



ADAPT-PT 2012

GETTING STARTED GUIDE

Contents

1	ADAPT-PT USER INTERFACE	6
2	ONE-WAY SLAB SUPPORTED ON BEAMS	7
2.1	GENERATE THE STRUCTURAL MODEL	9
2.1.1	Edit the project information	9
2.1.1.1	General Settings (Fig. 2.1-1)	9
2.1.1.2	Design Code (Fig. 2.1-2)	11
2.1.1.3	Design Settings (Fig. 2.1-3)	11
2.1.2	Edit the geometry of the structure	13
2.1.2.1	Enter Span Geometry (Fig. 2.1-4)	13
2.1.2.2	Enter Geometry Of Transverse Beam (Fig. 2.1-5)	15
2.1.2.3	Enter Supports-Geometry (Fig. 2.1-6)	16
2.1.2.4	Enter Supports Boundary Conditions (Fig. 2.1-7)	16
2.1.3	Enter Data	17
2.1.3.1	Enter the loading information (Fig. 2.1-8)	17
2.1.4	Edit the material properties	19
2.1.4.1	Enter The Properties Of Concrete (Fig. 2.1-10)	19
2.1.4.2	Enter The Properties Of Reinforcement (Fig. 2.1-11)	21
2.1.4.3	Enter The Post-Tensioning System Parameters (Fig. 2.1-12)	22
2.1.4.4	Edit Base Reinforcement (Fig. 2.1-13)	22
2.1.5	Edit the design criteria	23
2.1.5.1	Enter The Initial And Final Allowable Stresses. (Fig. 2.1-14)	23
2.1.5.2	Enter The Recommended Post-Tensioning Values (Fig. 2.1-15)	24
2.1.5.3	Select The Post-Tensioning Design Option (Fig. 2.1-16)	26
2.1.5.4	Specify The Tendon Profiles (Fig. 2.1-18)	27
2.1.5.5	Specify Minimum Covers For Post-Tensioning Tendons And Mild Steel Reinforcement (Fig. 2.1-19)	28
2.1.5.6	Specify Minimum Bar Length And Bar Extension Of Mild Steel Reinforcement (Fig. 2.1-20)	29
2.1.5.7	Input Load Combinations (Fig. 2.1-21, 22, 23)	30
2.2	SAVE AND EXECUTE THE INPUT DATA	32
2.3	CREATE REPORTS	38
3	COLUMN-SUPPORTED SLAB (TWO-WAY SYSTEM)	42
3.1	GENERATE THE STRUCTURAL MODEL	44
3.1.1	Edit the Project Information	44
3.1.1.1	General Settings (Fig. 3.1-1)	44
3.1.1.2	Design Code (Fig. 3.1-2)	45
3.1.1.3	Design Settings (Fig. 3.1-3)	45
3.1.2	Edit the Geometry of the Structure	47
3.1.2.1	Enter Span Geometry (Fig. 3.1-4)	47
3.1.2.2	Enter Support Geometry (Fig. 3.1-5)	48
3.1.2.3	Enter Support Boundary Conditions (Fig. 3.1-6)	49
3.1.3	Enter Data	50
3.1.3.1	Edit the Loading Information (Fig. 3.1-7)	50
3.1.4	Edit the Material Properties	52
3.1.4.1	Enter The Properties Of Concrete (Fig. 3.1-9)	52

3.1.4.2	Enter The Properties of Reinforcement (Fig. 3.1-10)	53
3.1.4.3	Enter The Post-Tensioning System Parameters (Fig. 3.1-11)	55
3.1.4.4	Edit Base Reinforcement (Fig. 3.1-12)	55
3.1.5	Edit the design criteria	56
3.1.5.1	Enter The Initial And Final Allowable Stresses. (Fig. 3.1-13)	56
3.1.5.2	Enter The Recommended Post-Tensioning Values (Fig. 3.1-14)	57
3.1.5.3	Select The Post-Tensioning Design Option (Fig. 3.1-15)	59
3.1.5.4	Specify The Tendon Profiles (Fig. 3.1-16)	59
3.1.5.5	Specify Minimum Covers For Post-Tensioning Tendons And Mild Steel Reinforcement (Fig. 3.1-17)	60
3.1.5.6	Specify Minimum Bar Length and Bar Extension of Mild Steel Reinforcement (Fig. 3.1-18)	61
3.1.5.7	Input Load Combinations (Fig. 3.1-19, 20, 21)	62
3.2	SAVE AND EXECUTE THE INPUT DATA	64
3.3	CREATE REPORTS	67
4	BEAM FRAME	70
4.1	Concrete:	70
4.2	GENERATE THE STRUCTURAL MODEL	72
4.2.1	Edit the project information	72
4.2.1.1	General Settings (Fig. 4.2-1)	72
4.2.1.2	Design Code (Fig. 4.2-2)	73
4.2.1.3	Design Settings (Fig. 4.2-3)	73
4.2.2	Edit the geometry of the structure	75
4.2.2.1	Enter Span Geometry (Fig. 4.2-4)	75
4.2.2.2	Enter Effective Flange Width (Fig. 4.2-5)	76
4.2.2.3	Enter Support Geometry (Fig. 4.2-6)	77
4.2.2.4	Enter Supports Boundary Conditions (Fig. 4.2-7)	77
4.2.3	Enter Data	78
4.2.3.1	Edit the loading information (Fig. 4.2-8)	78
4.2.4	Edit the material properties	80
4.2.4.1	Enter The Properties Of Concrete (Fig. 4.2-9)	80
4.2.4.2	Enter The Properties Of Reinforcement (Fig. 4.2-10)	80
4.2.4.3	Enter The Post-Tensioning System Parameters (Fig. 4.2-11)	82
4.2.4.4	Edit Base Reinforcement (Fig. 4.2-12)	82
4.2.5	Edit the design criteria	84
4.2.5.1	Enter The Initial And Final Allowable Stresses. (Fig. 4.2-13)	84
4.2.5.2	Enter The Recommended Post-Tensioning Values (Fig. 4.2-14)	84
4.2.5.3	Select The Post-Tensioning Design Option (Fig. 4.2-15, 16)	85
4.2.5.4	Specify The Tendon Profiles (Fig. 4.2-17)	86
4.2.5.5	Specify Minimum Covers For Post-Tensioning Tendons And Mild Steel Reinforcement (Fig. 4.2-18)	87
4.2.5.6	Specify Minimum Bar Length And Bar Extension Of Mild Steel Reinforcement (Fig. 4.2-19)	88
4.2.5.7	Input Load Combinations (Fig. 4.2-20)	89
4.3	SAVE AND EXECUTE THE INPUT DATA	91
4.4	CREATE REPORTS	97
5	NON-PRISMATIC (SEGMENTAL) COLUMN-SUPPORTED SLAB	100

5.1	Concrete:	100
5.2	Prestressing:	100
5.3	GENERATE THE STRUCTURAL MODEL	102
5.3.1	Edit the project information	102
5.3.1.1	General Settings (Fig. 5.3-1)	102
5.3.1.2	Design Code (Fig. 5.3-2)	103
5.3.1.3	Design Settings (Fig. 5.3-3)	103
5.3.2	Edit the geometry of the structure	105
5.3.2.1	Enter Span Geometry (Fig. 5.3-4 -14)	105
5.3.2.2	Enter Supports-Geometry (Fig. 5.3-15)	114
5.3.2.3	Enter Supports Boundary Conditions (Fig. 5.3-16)	115
5.3.3	Enter Data	115
5.3.3.1	Edit the loading information (Fig. 5.3-17)	115
5.3.4	Edit the material properties	117
5.3.4.1	Enter The Properties Of Concrete (Fig. 5.3-18)	117
5.3.4.2	Enter The Properties Of Reinforcement (Fig. 5.3-19)	117
5.3.4.3	Enter The post-tensioning system parameters (Fig. 5.3-20)	118
5.3.4.4	Edit Base Reinforcement (Fig. 5.3-21)	119
5.3.5	Edit the design criteria	119
5.3.5.1	Enter The Initial And Final Allowable Stresses (Fig. 5.3-22)	119
5.3.5.2	Enter The Recommended Post-Tensioning Values (Fig. 5.3-23)	120
5.3.5.3	Select The Post-Tensioning Design Option (Fig. 5.3-24)	121
5.3.5.4	Specify The Tendon Profiles (Fig. 5.3-25)	122
5.3.6	Specify Minimum Covers For Post-Tensioning Tendons And Mild Steel Reinforcement (Fig. 5.3-26)	123
5.3.6.1	Specify Minimum Bar Length and Bar Extension Of Mild Steel Reinforcement (Fig. 5.3-27)	123
5.3.6.2	Input Load Combinations (Fig. 5.3-28, 29, 30)	124
5.4	SAVE AND EXECUTE THE INPUT DATA	126
5.5	CREATE REPORTS	130

Getting Started

Thank you for choosing ADAPT-PT 2012 as your post-tensioned concrete design software. This getting started guide is intended to provide you with a brief overview of the capabilities and functionality of this product. In following through the examples below, you will learn how to model a structure and perform the necessary design steps. For more details about the program please refer to the ADAPT-PT user manuals. These manuals are located at our website www.adaptsoft.com, our support website www.adaptsolutions.wordpress.com or can be requested from our support team at support@adaptsoft.com.

1 ADAPT-PT USER INTERFACE

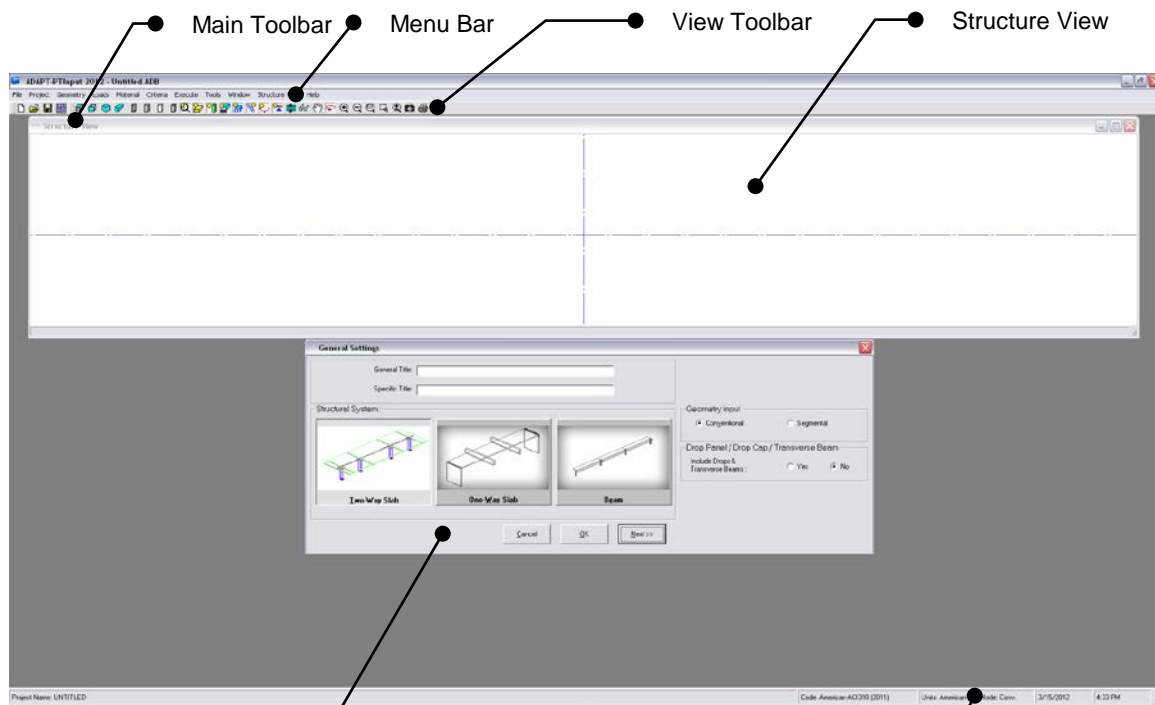


FIGURE 1: ADAPT-PT User Interface

Input Forms

Status Bar

The model is created with the help of a wizard which consists of various *Input Forms*. Each *Input Form* can be accessed at any time through the *Menu Bar*. The user-defined input on the *Input Forms* is displayed real-time in the *Structure View*. The view of the structure can be modified with the help of the *View Toolbar* which contains *View Tools* allowing you to change the perspective, turn on and off components and zoom. The *Main Toolbar* contains *Common Tools* such as *New Project*, *Open Project* and *Save Project*.

2 ONE-WAY SLAB SUPPORTED ON BEAMS¹

The objective of this example is to demonstrate the step-by-step procedure of ADAPT-PT to analyze and design a post-tensioned continuous slab supported on parallel beams or walls. A typical six-span, one-way slab supported on beams, as occurs in parking structures is selected. The example covers the following aspects of the program:

- Modeling of one-way slabs spanning between parallel beams;
- Application of effective force method in design of post-tensioned members;
- Optimization of design through selection of different post-tensioning forces in different spans; and
- Application of unbonded tendons.

The geometry of the structure is shown in **Figure 2-1**². Other characteristics of the structure and the design requirements are given in the following.

Six-span, one-way slab spanning 18'-0 (5.49 m) between 14 in. x 34 in. (356mm x 864 mm) cast in place concrete beams.

Thickness of slab = 5 in (127 mm)

(i) Material Properties

○ Concrete:		
Compressive strength, f'_c	= 4000 psi	(27.58 MPa)
Weight	= 150 pcf	(2403 kg/m ³)
Modulus of Elasticity	= 3604 ksi	(24849 MPa)
Age of Concrete at stressing	= 3 days	
Compressive strength at stressing, f'_{ci}	= 3000 psi	(20.68 MPa)

¹ Copyright ADAPT Corporation 2012

² The geometry, loading, material properties and the design criteria selected are the same as those in PTI's publication for Design of Post-Tensioned Slabs. The results, however, are somewhat different, since in PTI's publication only part of the forces from post-tensioning are accounted for. .

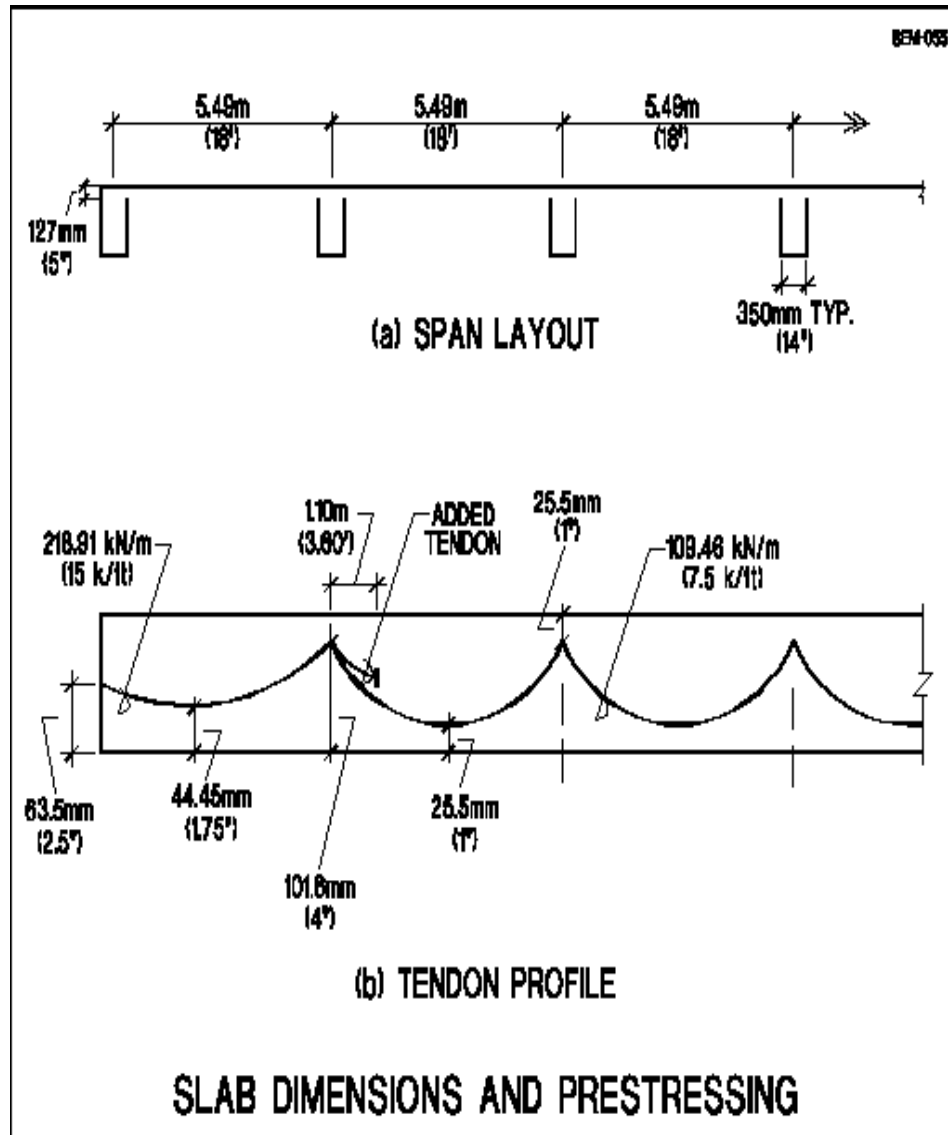


FIGURE 2-1

- Prestressing:
 - Low Relaxation, Unbonded System
 - Strand Diameter = ½ in (13 mm)
 - Strand Area = 0.153 in² (98 mm²)
 - Modulus of Elasticity = 28000 ksi (193054 MPa)
 - Coefficient of angular friction, μ = 0.07
 - Coefficient of wobble friction, K = 0.0014 rad/ft (0.0046 rad/m)
 - Specified Ultimate strength of strand, fpu = 270 ksi (1862MPa)
 - Ratio of jacking stress to strand's ultimate strength = 0.8
 - Anchor set = 0.25 in (6.35 mm)

Minimum strand cover			
From top fiber		= 0.75 in all spans (19.05 mm)	
From bottom fiber			
	Interior spans	= 0.75 in	(19.05 mm)
	Exterior spans	= 1.5 in	(38.1 mm)
○ Non-prestressed Reinforcement:			
Specified yield stress f_y		= 60 ksi	(413.69 MPa)
Modulus of Elasticity		= 29000 ksi	(199,949 MPa)
Minimum Rebar Cover		= 1 in	(25.4 mm)
(ii) Loading:			
Dead load		= self weight + 5 psf (allowance for curbs, lighting, drainage etc)	
		= (5/12) * 150 + 5	
		= 68 psf	(3.26 kN/m ²)
Live load		= 50 psf	(2.39 kN/m ²)

2.1 GENERATE THE STRUCTURAL MODEL

2.1.1 Edit the project information

2.1.1.1 General Settings (Fig. 2.1-1)

Under *Options* in the main menu, select **American** for the *System of Units*.

Open a new project by clicking either **New** on the *file* menu or the **New Project** button on the toolbar. This automatically opens the *General Settings* input screen as shown in **Fig. 2.1-1**. You can enter the General title and/or Specific title of the project.

For the purpose of this example, enter the *General title* as **Six-Span One-Way Slab**. This will appear at the top of the first page of the output. Enter the *Specific title* as **Example 1**. This will appear at the top of each subsequent page of the output.

Next, select the *Structural System* as **One-Way Slab**. For the option to include Transverse Beam, select **Yes**.

Next, select *Geometry Input* as **Conventional**. Segmental input is used for entering non-prismatic structures, i.e., those where the tributary width or the depth of the section changes within a span.

Click **Next** on the bottom line of this screen to open the input screen, *Criteria – Design Code*.

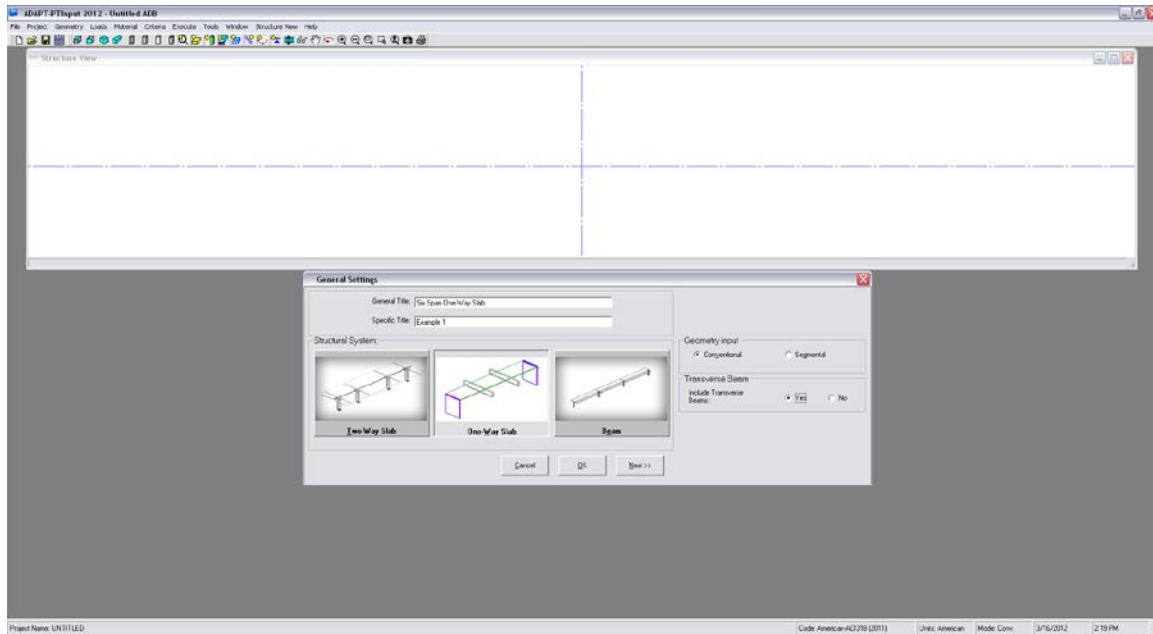


FIGURE 2.1-1

2.1.1.2 Design Code (Fig. 2.1-2)

New! In the second step you can specify the design code. In the Design Code screen, set the code as **ACI318-2011/IBC 2012**. Note that this is the single new code added to PT 2012.

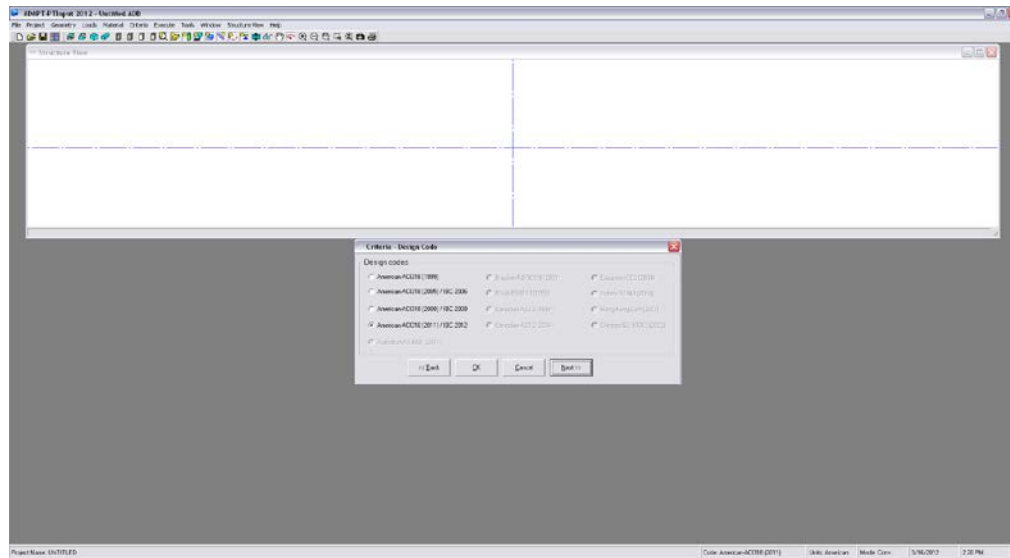


FIGURE 2.1-2

2.1.1.3 Design Settings (Fig. 2.1-3)

This screen is divided in two parts: *Analysis options* and *Design options*.

In *Analysis options*, you can select various calculation settings.

First, select the execution mode as interactive. In this mode, you have the opportunity to optimize the design by adjusting the tendon forces and tendon drapes in each span in the Recycle window. This will be explained later in this section.

Next, select **Yes** for the *Reduce Moments to Face- of- Support* option. This option indicates that the calculated centerline moments, at each support are, adjusted to the face-of support. In addition to the centerline moments, ADAPT-PT prints out the moments reduced to the face-of- support.

For a one-way slab system, the Equivalent Frame method is not applicable.

Select **No** for the option to *Redistribute moments*.

Then, there is an option to *Increase Moment of Inertia Over Support*. This option accounts for the increased stiffness of a slab over its support. In this example the slab is supported on beams. The program correctly accounts for the added

stiffness over the beam support. For beam supported slabs **No** is applicable. For wall supported slabs **Yes** will apply. For this example select **No**.

In *Design options*, you can either *Use all provisions of the code* that you selected in the previous step, or *Disregard the following provisions* such as *Minimum rebar for serviceability*, *Design capacity exceeding cracking moment*, and *Contribution of prestressing in strength check*.

The option to *Include (DL + 0.25LL) case of UBC* allows you to enter the *Ratio of reduced live load to actual live load*. Leave this option unchecked for this example. This is a UBC (Uniform Building Code) requirement (not required by ACI 2011 and IBC 2009) used to determine the amount of mild steel reinforcement.

Next is the option to *Generate moment capacity based on* **Design Values** or **User-Entered Values**. If **Design Values** is selected, the program will calculate and report positive and negative moment capacities based on prestressing steel, base reinforcement as defined by the user (this is discussed later in this section) and program-calculated reinforcement. Demand moments at 20th points along each span are also reported. When **User-Entered Values** is selected, the program will calculate and report similar moment capacities and demand moments where the capacities are based on prestressing steel and based reinforcement as defined by the user. For this example, select **Design Values**.

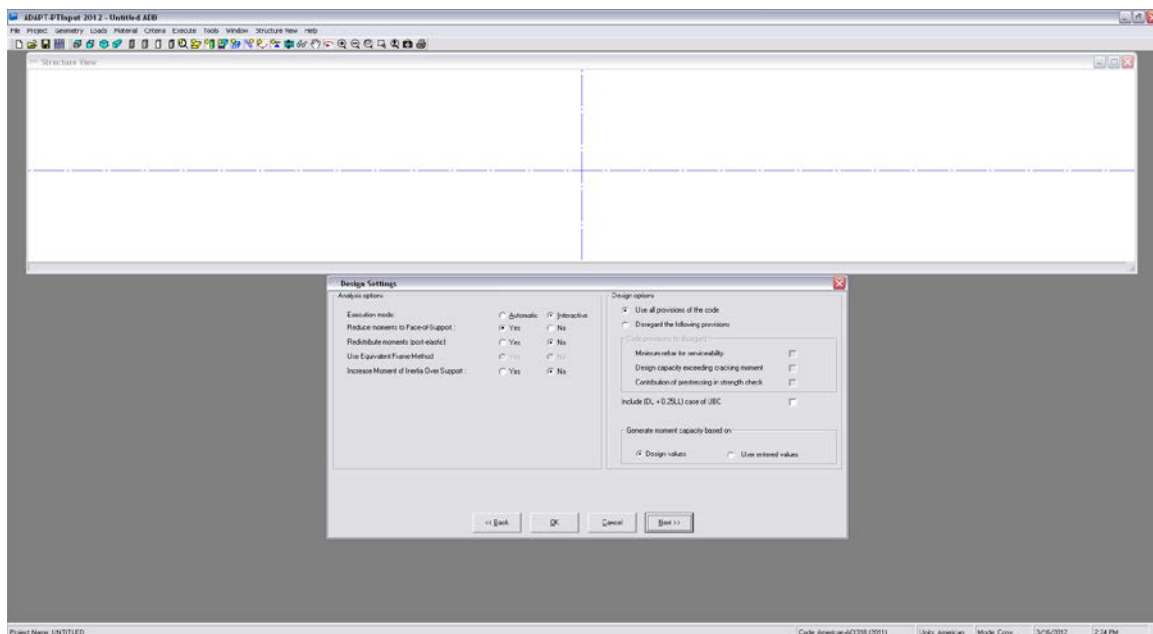


FIGURE 2.1-3

Click **Next** at the bottom right of the *Design Settings* screen to open the *Span Geometry* input screen.

2.1.2 Edit the geometry of the structure

2.1.2.1 Enter Span Geometry (Fig. 2.1-4)

This screen is used to enter the cross-sectional geometry of the slab at mid-span.

Set the *Number of Spans* as **6** either by clicking the **up arrow** or using **CTRL +**.

Next, enter the dimensions. All dimensions are defined in the legend at the top of the screen and/or are illustrated in the appropriate section FIGURE. The section type for any span can be changed by clicking on the button in the *Sec* (Section) column.

Select the section, *Sec*, as **Rectangular** and edit **18 ft** (5.49 m) for length, *L*, **12** in (305 mm) for width, *b*, and **5** in (127 mm) for height, *h*, for all spans. To enter typical values, type the **value** into the appropriate cell in the top row and then press **enter**. The typical value will be copied to all the spans.

As you enter the values, the span is displayed in real-time in the 3D window. You can zoom in and out in the *Structure View* with the help of your mouse wheel or with the help of the *Zoom In* or *Zoom out* buttons in the *View Toolbar*.

You can access special data editing options by selecting data cells and right clicking. Available options include Insert New Line, Delete Line, Copy Selected Lines, and Paste Lines.

The Reference height (*Rh*) identifies the position of a reference line from which the position of the tendon's centroid (CGS) is measured. Typically, the reference height is set equal to the slab depth. This will result in the tendon height being specified as positive values up from the soffit of slab. The Reference height (*Rh*) is also used to create steps in the slab. Click the **?** button with the **Rh** definition in the legend to learn more.

Type the Reference height, *Rh*, as **5 inches**(127 mm), i.e., slab depth, for all spans.

The Left and Right Multiplier columns (<-M and M->) are used to specify the tributary width, to indicate how much of the tributary falls on either side of the frame line. Tributary widths can be specified using either the Unit Strip method or the Tributary method. For this example, Tributary method is used, i.e., total tributary (**12 inches** (305 mm)) is entered as the *b* dimension. Also, equal tributary falls on either side of the frame line, so enter **0.50** for both left and right multiplier. In general, it is better to select a larger multiplier for the tributary, i.e. 5 on each side. This will result in a more rational selection of reinforcing bars.

However, regardless of the selection of tributary value, the program reports the correct stresses and the required rebar areas.

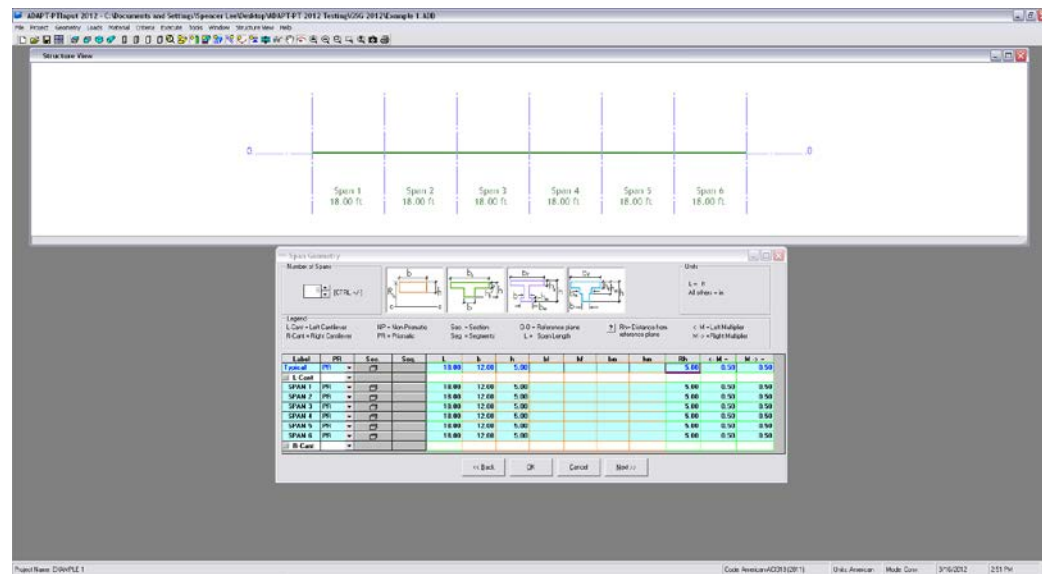


FIGURE 2.1-4

Click **Next** on the bottom line to open the next input screen, *Geometry-Transverse Beam*.

2.1.2.2 Enter Geometry Of Transverse Beam (Fig. 2.1-5)

Note: The screen display will vary depending on which structural system has been specified in the *Design Setting* screen.

Next, enter the dimensions of *Transverse Beam*. Type **34** inches (864 mm) for the height, *H*, and **7** inches (178 mm) for the width on the left side of the centerline (D1), and **7** inches (178 mm) for the width on the right side of the centerline (D2) for all spans. Note, you cannot type a dimension for the first span of column D1 and for the last span of column D2, since the span length is a center-to-center dimension.

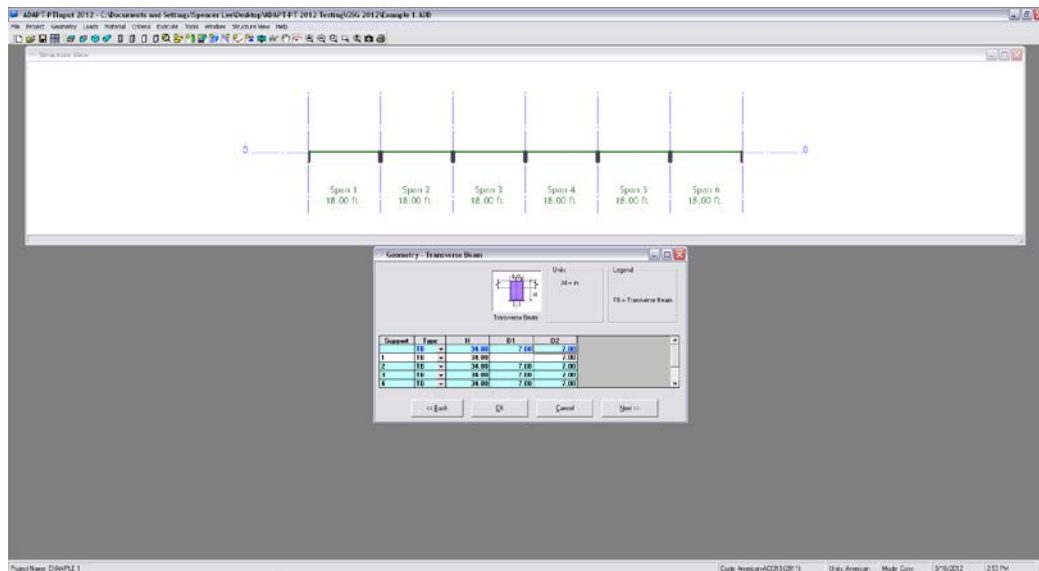


FIGURE 2.1-5

Click on **Next** on the bottom line to open the *Drop Panel Geometry* screen.

As there are no drop panels defined in this example, click on **Next** on the bottom of the screen to open the *Supports Geometry* input screen.

2.1.2.3 Enter Supports-Geometry (Fig. 2.1-6)

This screen is used to input column or wall heights, widths and depths. You may enter dimensions for columns/walls above and/or below the slab.

Select **Point Support or Transverse Beam** from the *Support selection* since the slab is supported by beams. The program will automatically consider the beams as a point support.

On this input screen, you can select for each support whether the left edge and the right edge of that support is interior or exterior. This option only applies to two-way slabs and defines how a column is treated for punching shear in the direction perpendicular to span length. This is applied later in the two-way slab design example.

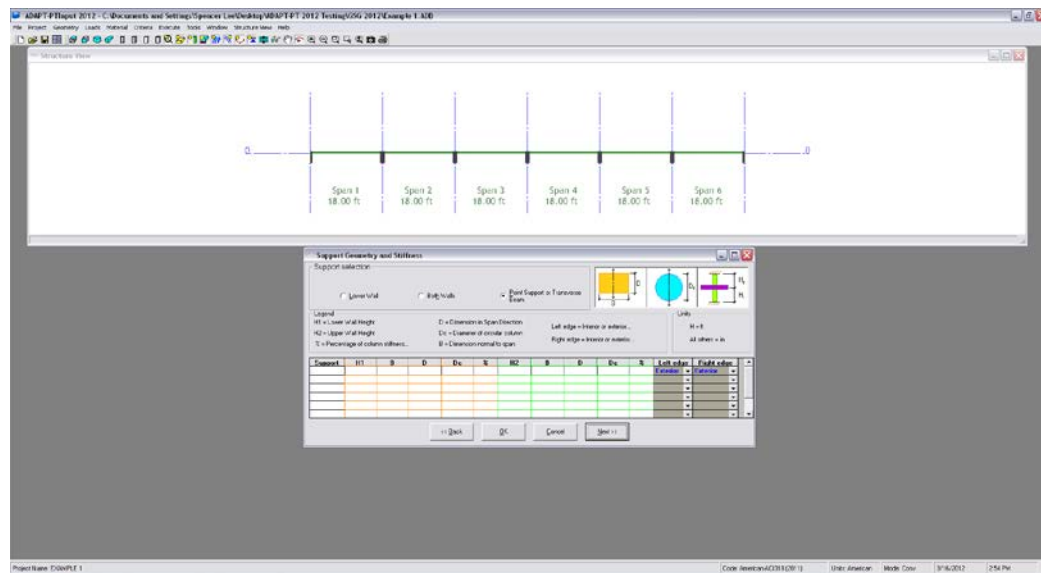


FIGURE 2.1-6

Click **Next** on the bottom line to open the *Supports-Boundary Conditions* input screen.

2.1.2.4 Enter Supports Boundary Conditions (Fig. 2.1-7)

This screen is used to enter the support widths and column boundary conditions.

Type **14** inches (356 mm) (the width of the beam) for *SW* (Support Width) in the typical row and press **enter**. This input value will be used to calculate the reduced moments. There are no columns or walls selected for this example. This deactivates the input for lower and upper column/wall boundary conditions.

You could also check **SW = Actual width of support** as the beams were defined with a width of 14 inches. Note that if this option is used, the program will not allow the user to manually enter a value for support width.

Leave the *End Support Fixity* for both left and right support as the default **No**. This will be used when the slab or beam is attached to a fully-fixed or rigid member such as a wall.

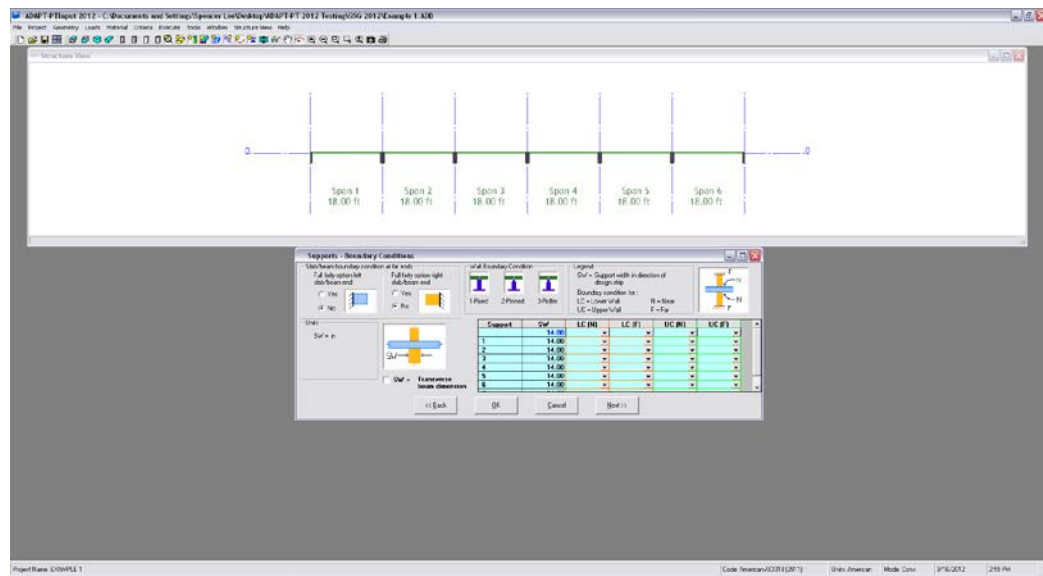


FIGURE 2.1-7

Click **Next** at the bottom of the screen to open the next input screen, *Loading*.

2.1.3 Enter Data

2.1.3.1 Enter the loading information (Fig. 2.1-8)

Any number of different loads and load types may be entered for a span.

Load types available in PT 2012 are: *Uniform, Partial Uniform, Concentrated, Moment (Concentrated), Line, Triangular, Variable and Trapezoidal*.

Enter the span number as **1** in the *Span* column. If the loads are the same for all the spans, you can type **ALL** or **all** in the *Span* column. This will copy the data to all the spans.

If you choose not to include Self-weight, the self-weight (**SW**) can be input in its own load *Class*. In any case, you can choose to specify additional dead load as a superimposed dead load (**SDL**).

PT 2012 gives you the option to specify any other load (i.e. hydrostatic, soil, etc.) in the **X Class** loading.

Select the *Class* as **SDL** from the drop down list and specify the load type as uniform, either by typing **U** in *L-?* or by **dragging the icon** from the graphics of the uniform loading. The default load type for the load class is **L-U**; this will be used for this tutorial.

Type **0.01 k/ft²** (0.239 kN/m²)(without self-weight) for superimposed dead load in the *w* column. You can enter DL with or without self-weight, since the program can calculate self-weight automatically. In order to calculate the self-weight automatically, you must select **Yes** to the *Include Self-Weight* in the option box to the top right of the screen and enter a unit weight of concrete. The program defaults to **150 pcf** (2402.85 kg/m³) as the unit weight. Other values can be input to overwrite this value.

Repeat the procedure by changing *Class* to **LL** and the *w* value to **0.05 k/ft²** (2.39 kN/m²).

You can either enter the **SDL** and the **LL** for each span (Figure 2.1-8), or you can type *all* and define SDL and LL once for all spans. The loads then get applied to all spans (Figure 2.1-9).

Select **Yes** for *Skip Live Load?* at the top left of the screen and enter the *Skip Factor* as 1.

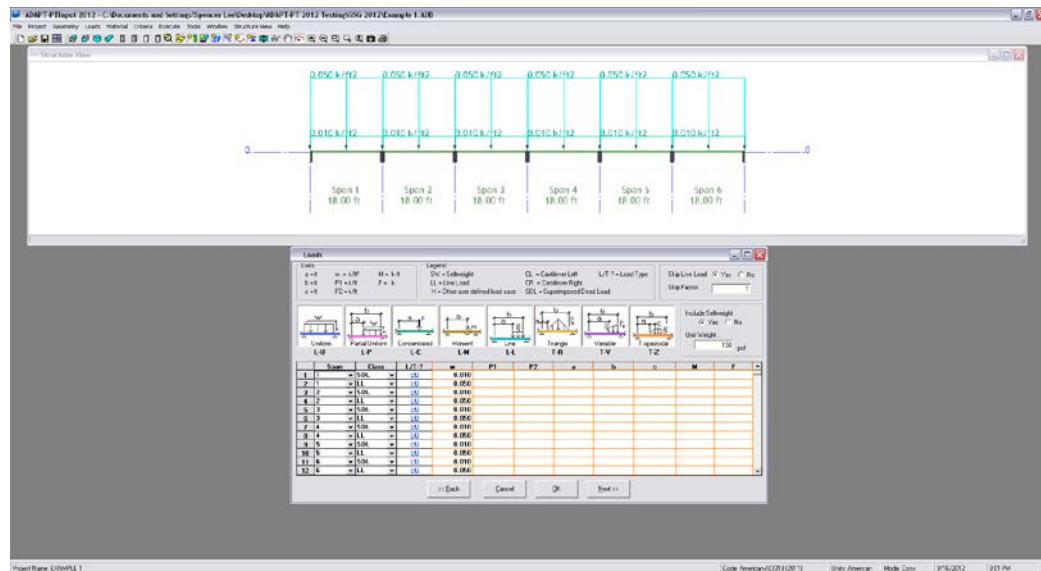


FIGURE 2.1-8

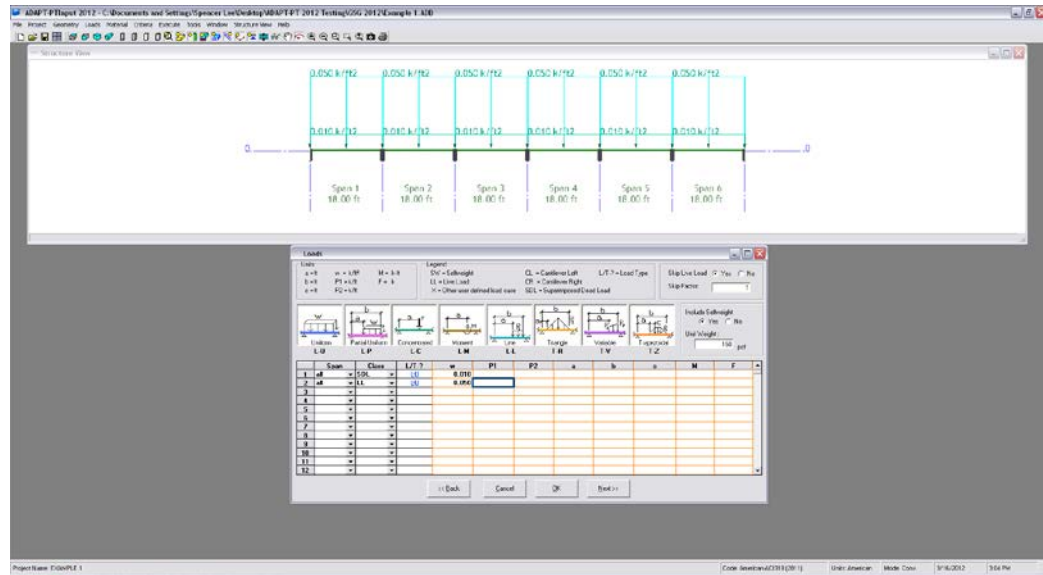


FIGURE 2.1-9

Click **Next** at the bottom of the screen to open the *Material-Concrete* input screen.

2.1.4 Edit the material properties

2.1.4.1 Enter The Properties Of Concrete (Fig. 2.1-10)

Select the **Normal Weight** and enter the *Strength at 28 days* as **4000** psi (27.58 MPa) for slab/beam. When you press **enter** for the strength input value, the modulus of elasticity will calculate automatically based on the concrete strength and the appropriate code formula. For this tutorial, keep the **default values** of strength and creep coefficient. Creep coefficient will be used in the calculation of long-term deflection.

Enter the *Concrete strength at prestressing (initial condition)* as **3000** psi (20.68 MPa).

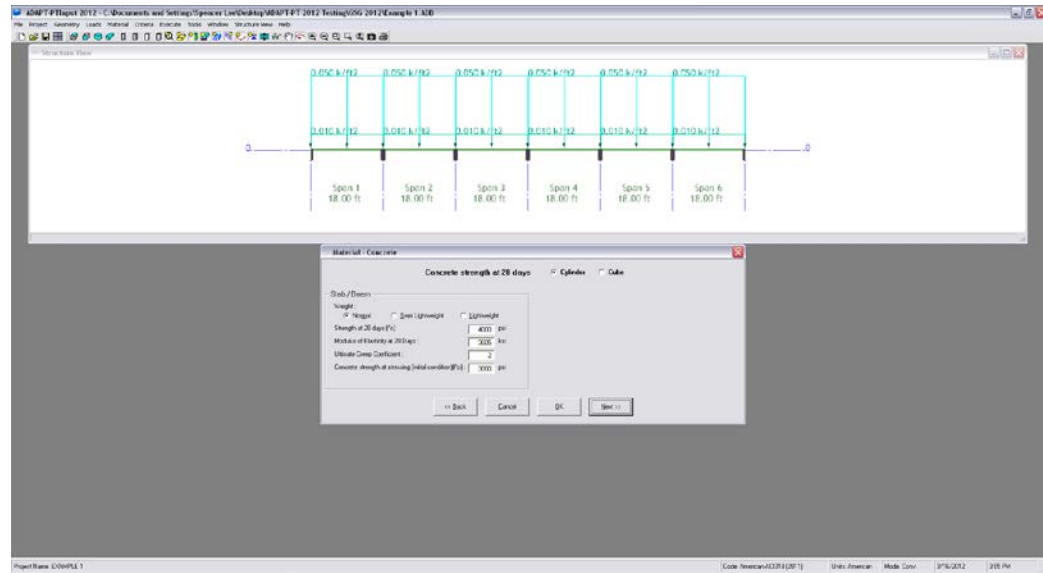


FIGURE 2.1-10

Click **Next** at the bottom of the screen to open next input screen *Material Reinforcement*.

2.1.4.2 Enter The Properties Of Reinforcement (Fig. 2.1-11)

The screen is divided into two parts: *Longitudinal reinforcement* and *Shear reinforcement*.

In the section *Longitudinal reinforcement*, keep the default values for *Yield Strength* and *Modulus of Elasticity*. Change the *Preferred Bar Sizes for Top and Bottom* to **5** and **6** respectively (16, 19). This will be used when calculating the number of bars required.

In Shear reinforcement, keep the default *Preferred Stirrup Bar Size* and the *Yield strength shear reinforcement* and the *Number of legs*.

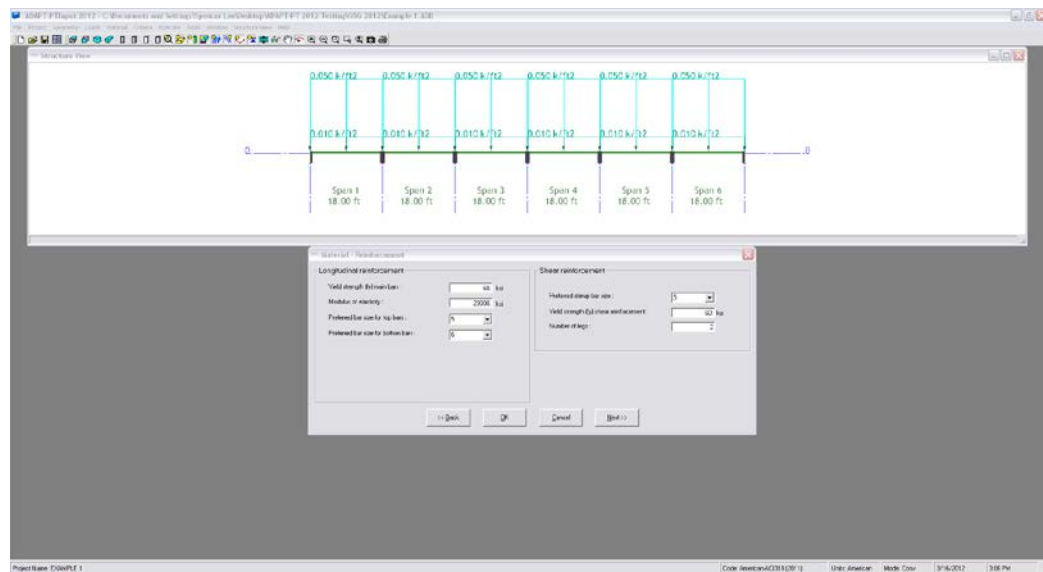


FIGURE 2.1-11

Click **Next** at the bottom of the screen to open up next screen, *Material Post-Tensioning*.

2.1.4.3 Enter The Post-Tensioning System Parameters (Fig. 2.1-12)

Select the *Post-tensioning system* as **Unbonded** and leave the default values of the other properties as they are.

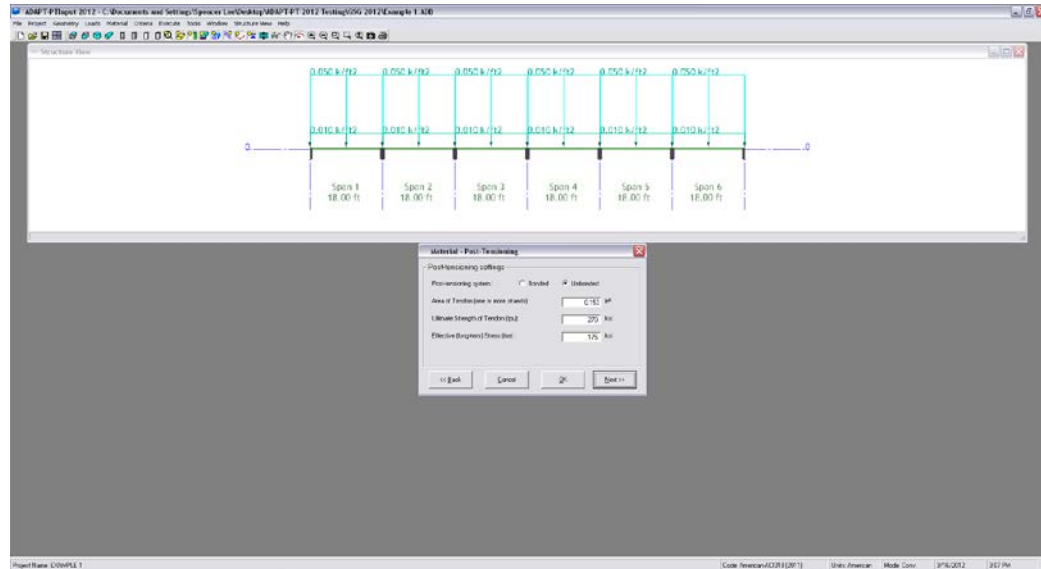


FIGURE 2.1-12

Click **Next** at the bottom of the screen to open the input screen, *Base Non-Prestressed Reinforcement*.

2.1.4.4 Edit Base Reinforcement (Fig. 2.1-13)

The program allows you to specify a base reinforcement that is taken into consideration when designing the structure. Select **Yes** in the *Base Reinforcement* section.

You have the choice between defining a mesh or isolated rebar. For this example choose **Isolated** from the drop down box.

Next, specify the span where your base reinforcement starts. For this example, let the rebar start at the beginning of span 1. Therefore, enter a **1** in *First end location* and a **0** in *X1/L*.

If you wanted to let the rebar start mid-span of span 1, you could enter 0.5 for *X1/L*. For this example, however, we keep **0**.

To specify the end of the reinforcement at the end of span number 6, define **6** for *Second end location* and **1** for *X2/L*.

Furthermore, you specify **4** bars (*Number*) with *Bar Size* of **6** as **Bottom** bars with a *Cover* of **2** inches.

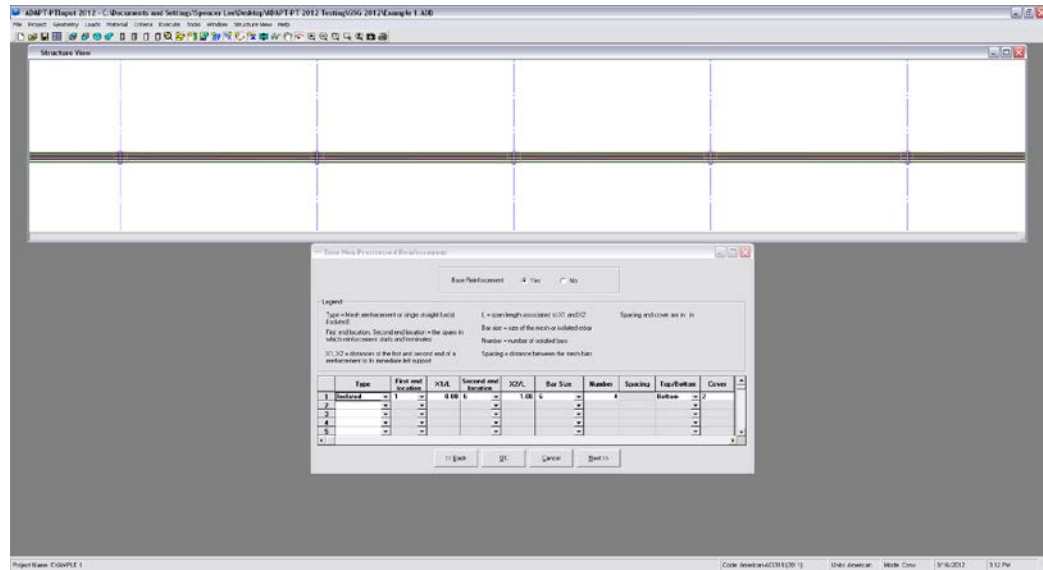


FIGURE 2.1-13

Click **Next** at the bottom of the screen to open the input screen, *Criteria – Allowable Stresses*.

2.1.5 Edit the design criteria

2.1.5.1 Enter The Initial And Final Allowable Stresses. (Fig. 2.1-14)

Tension stresses are input as a multiple of the square root of f'_c , and compressive stresses are input as a multiple of f'_c .

The default values given in this screen are according to the appropriate code, i.e., according to ACI 318-2011 for this case. So leave it as is.

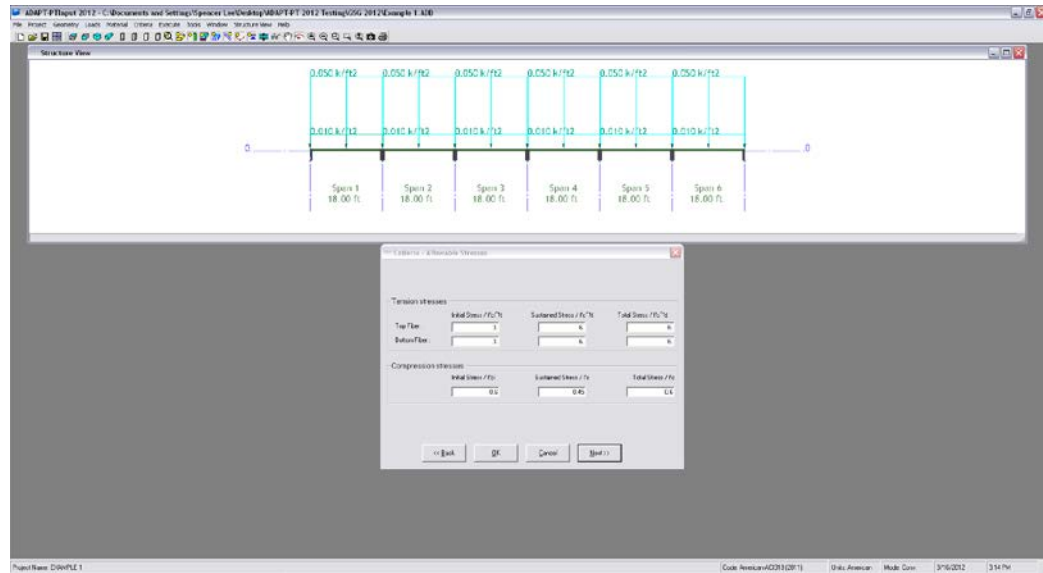


FIGURE 2.1-14

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Recommended Post-Tensioning Values*.

2.1.5.2 Enter The Recommended Post-Tensioning Values (Fig. 2.1-15)

This screen is used to specify minimum and maximum values for average precompression (P/A : total prestressing divided by gross cross-sectional area) and percentage of dead load to balance (W_{bal}). These values are used by the program to determine the post-tensioning requirements and the status of the P_{min}/P_{max} and W_{BAL} Min/ Max indicators on the Recycle window.

The values given as default are according to the code and the experience of economical design. Therefore, keep the default values.

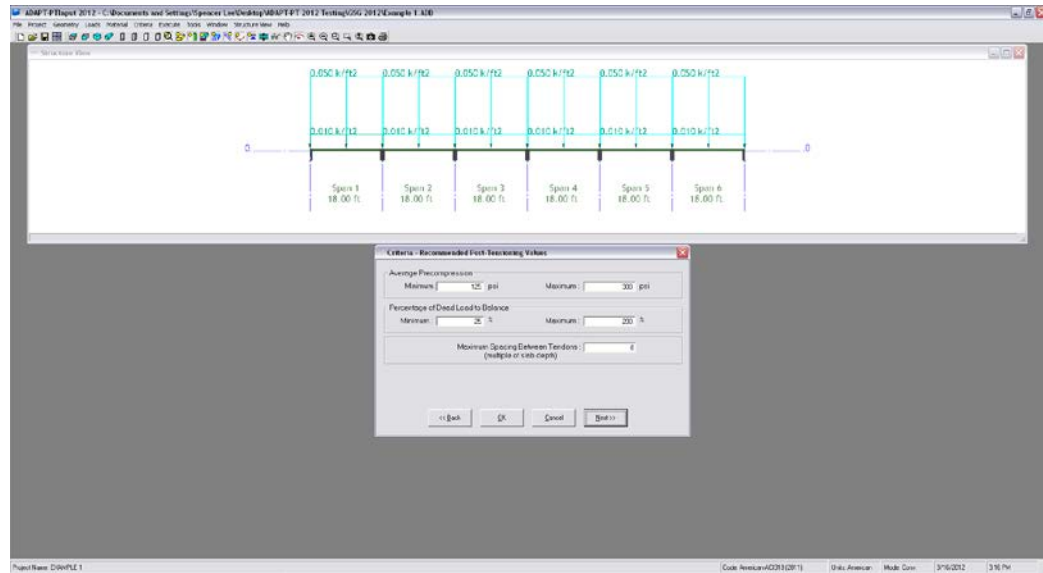


FIGURE 2.1-15

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Calculation Options*.

2.1.5.3 Select The Post-Tensioning Design Option (Fig. 2.1-16)

The two design options are “Force selection” and “Force/Tendon Selection” as shown in **Figure 2.1-16**. “Force Selection” is the default option.

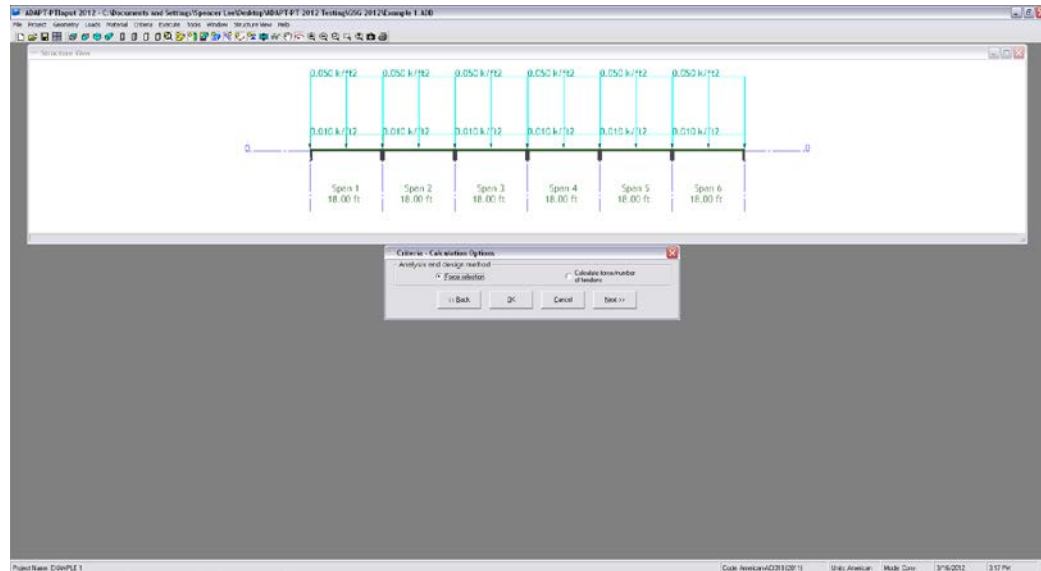


FIGURE 2.1-16

Select **Force/ Tendon Selection** option, then the screen will prompt for the information required to calculate the prestress losses as shown in **Figure 2.1-17**.

Enter the information given in the material properties to calculate friction losses as shown in **Figure 2.1-17**.

Long-term losses may either be entered as a lump sum value or the program can calculate them using the provided information. Select **Yes** to *Perform Long-Term Loss Calculation*. Enter *Age of stressing* as **3** days and press **Enter**. Then the *Strength* and *Modulus of Elasticity at Stressing* will be calculated automatically by the program. However, if concrete strength at stressing is established through cylinder/cube tests, enter the test result. For most anchorage devices, there is a specified minimum concrete strength for stressing. In this tutorial, the minimum value is 3000 psi (20.68 MPa). So, enter **3000** psi (20.68 MPa) for *strength of concrete at stressing*.

Answer **Yes** to *Are All Tendons Stressed at One time?* question. This information is used to determine the stress losses in prestressing due to elastic shortening of the member.

Use **80%** for *Relative Ambient Humidity (RH)* and **2.5** inches (64 mm) for *Volume to Surface Ratio (V/S)*. V/S is the calculated value from the given section dimensions. Specify that *All tendons are stressed at the same time*.

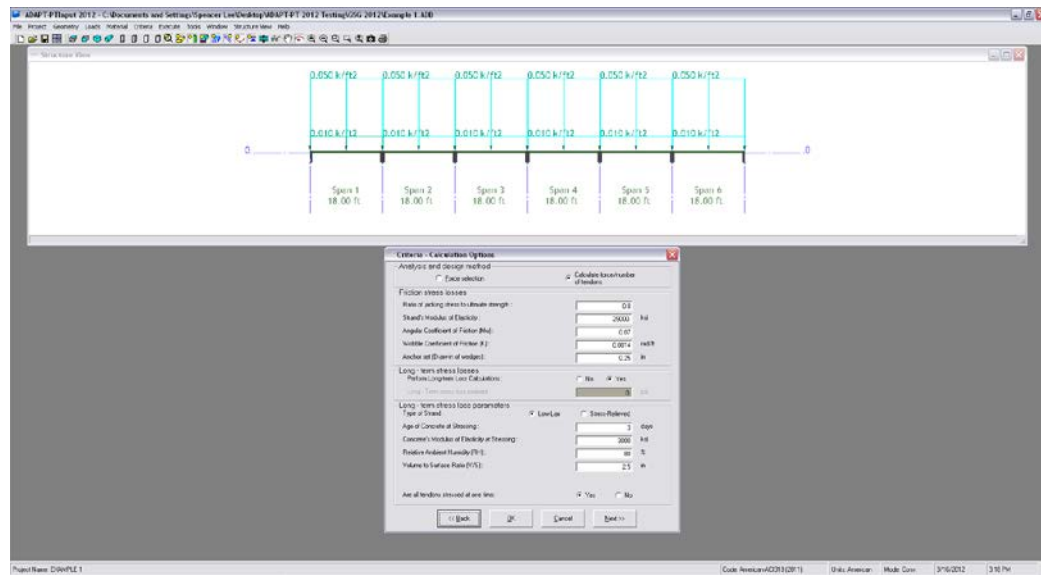


FIGURE 2.1-17

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Tendon Profile* .

2.1.5.4 Specify The Tendon Profiles (Fig. 2.1-18)

The program allows you to specify up to three tendon paths per span. You can define one profile for each of the three tendons.

In the section *Option for tendons* you can define the *Default extension of terminated tendon as fraction of span*.

Also, you can specify the *Shape of tendon extension* from the *Left end* and the *Right end*.

For this example, leave the default values.

In this example we only use tendon A. From the *Type* drop down list, select **1** for Reversed parabola and change the inflection points ($X1/L$ and $X3/L$) to **zero**, since we assumed a parabola with no inflection points. Keep the low point ($X2/L$) at mid span, i.e., at **0.5**, except for the first and last spans. From the calculation, the low point for the first and last spans is at $0.366 \cdot L$ and $0.634 \cdot L$, respectively from the left support. So enter $X2/L$ for the first and last spans as **0.366** and **0.634**, respectively.

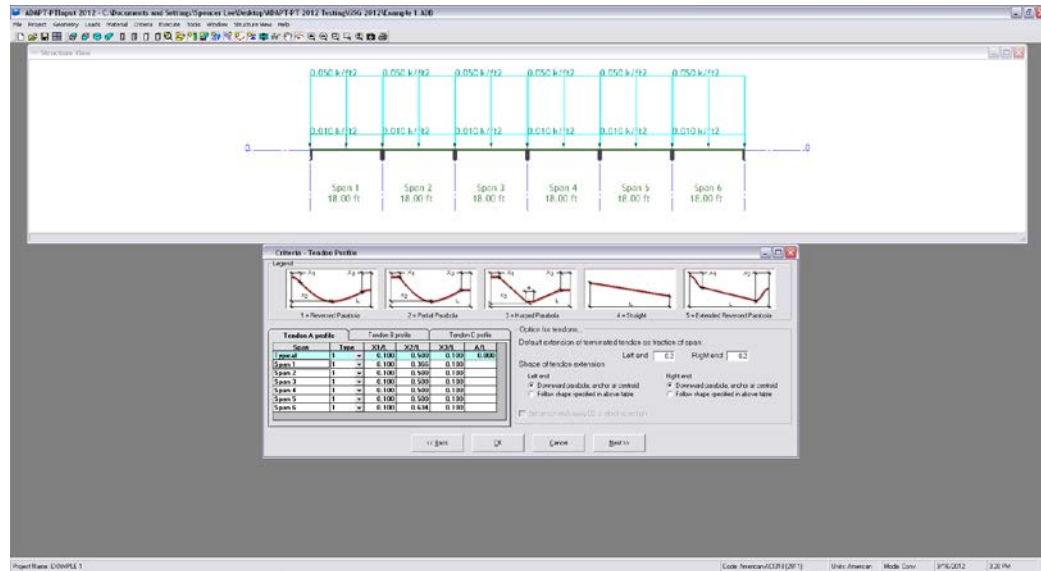


FIGURE 2.1-18

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Minimum Covers*.

2.1.5.5 Specify Minimum Covers For Post-Tensioning Tendons And Mild Steel Reinforcement (Fig. 2.1-19)

The concrete cover for the pre-stressing steel is determined as a distance from the extreme fiber to the center of gravity of the strand (CGS). Therefore, for ½ inch (13 mm) strand, CGS is minimum cover + ½ * ½ i.e., $cgs = cover + 0.25$ ($cgs = cover + ½ * 13$).

Edit **1** in (25 mm) *cgs* for both the top fiber and the interior spans of the bottom fiber. Use **1.75** inches (44 mm) for the *exterior span* for the bottom fiber.

Edit **1** in (25 mm) *Cover* for both top and bottom non-prestressed reinforcement.

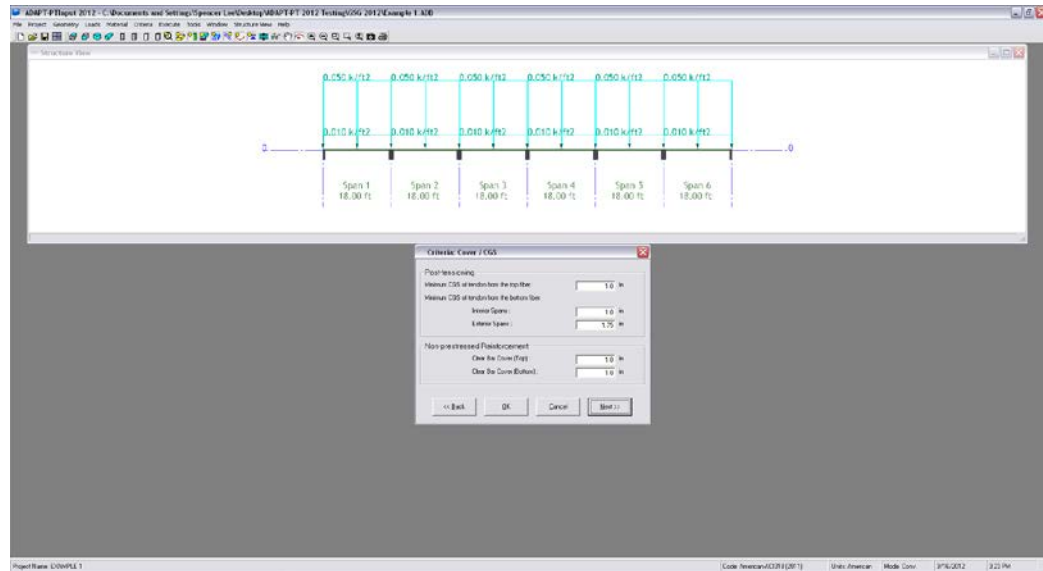


FIGURE 2.1-19

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Minimum Bar Extension*.

2.1.5.6 Specify Minimum Bar Length And Bar Extension Of Mild Steel Reinforcement (Fig. 2.1-20)

The values given as default are according to the appropriate code, for this tutorial according to ACI 318-2011 code. Therefore keep the default values as is.

The values entered for cut-off lengths are used to calculate top and bottom bar lengths when minimum reinforcement requirements govern.

The development length of reinforcement required for strength will extend the reinforcement by the given value beyond the calculated length. Please note that the program does not calculate this value automatically per bar size used.

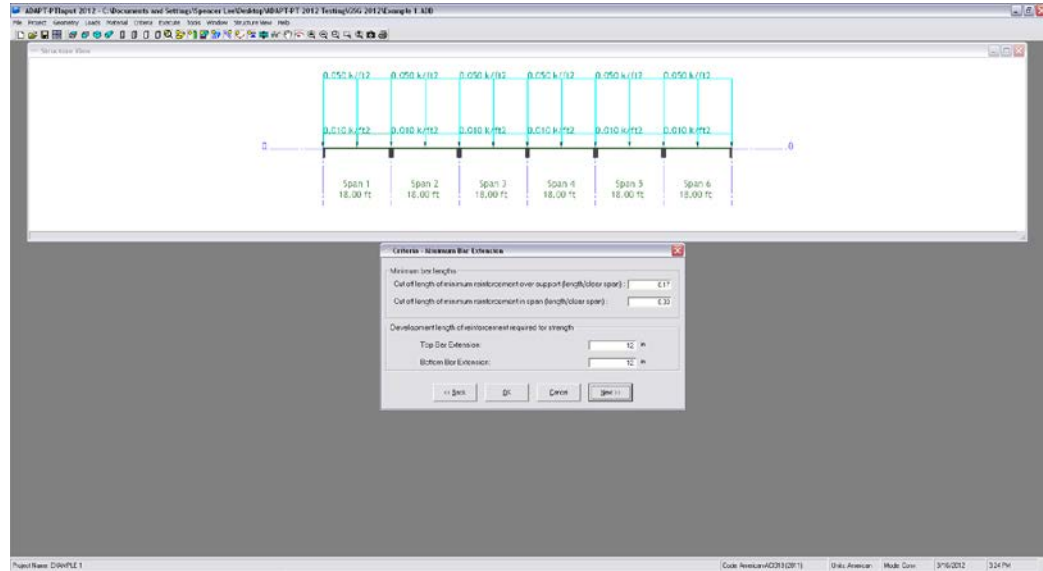


FIGURE 2.1-20

Click **Next** at the bottom of the screen to open the next input screen, *Load Combinations*.

2.1.5.7 Input Load Combinations (Fig. 2.1-21, 22, 23)

Figure 2.1-23 shows the screen which is used to input the load combination factors for service and strength (ultimate) load conditions. It is also used to enter any applicable strength reduction factors.

The program allows you to specify four strength load combinations and four service load combinations. For ACI 318-2011, two of the service load combinations are reserved for sustained load and two for total load.

New! Check the check mark to *Include lateral loads* and click on the *Set Values* button to define *Lateral moments* (**Fig. 2.1-21**) and *Lateral load combinations* (**Fig. 2.1-22**). ADAPT-PT 2012 now defaults to code prescribed load combinations including lateral loading. For this example, do not include lateral loads.

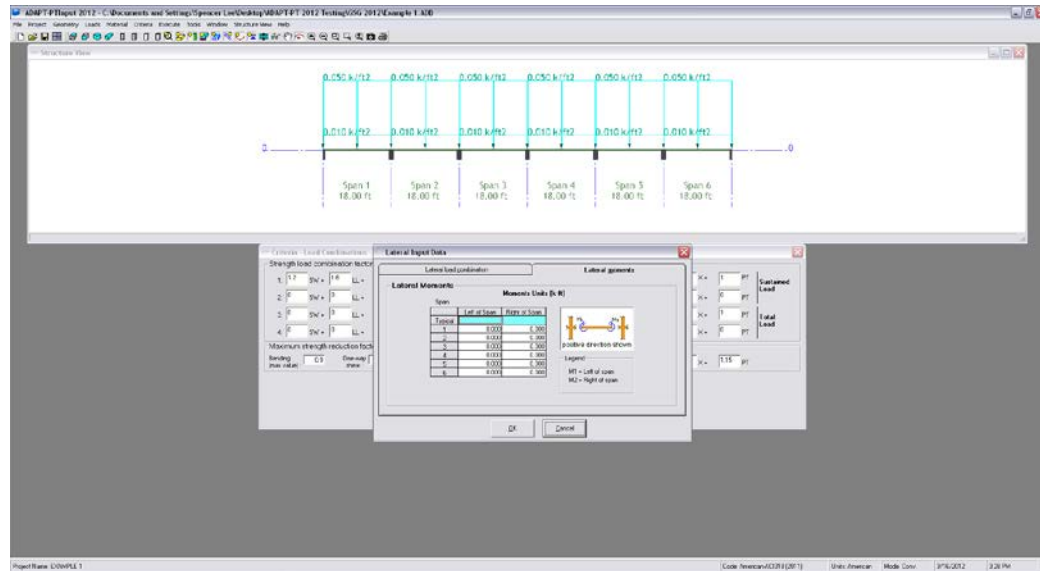


FIGURE 2.1-21

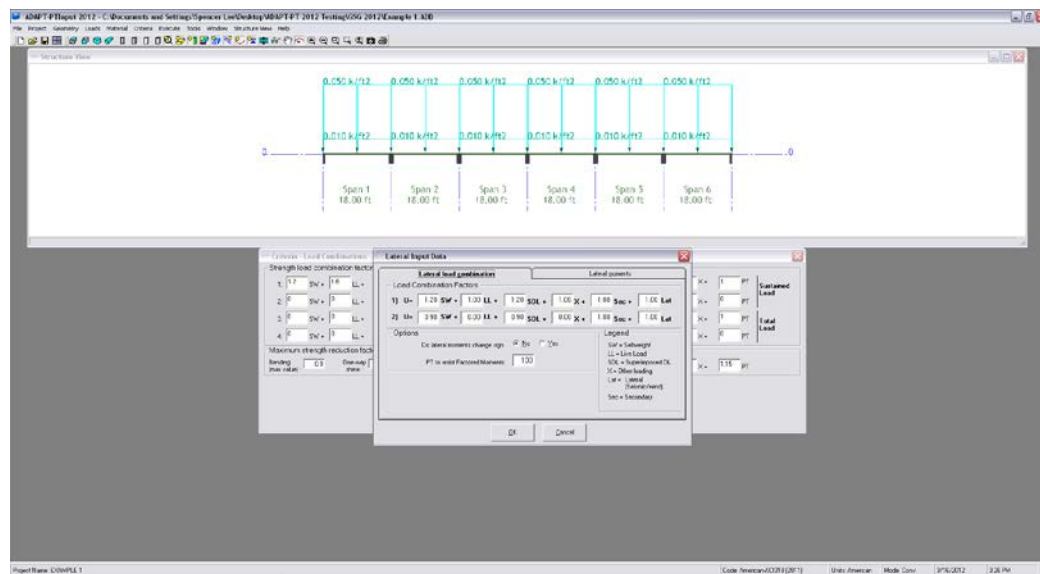


FIGURE 2.1-22

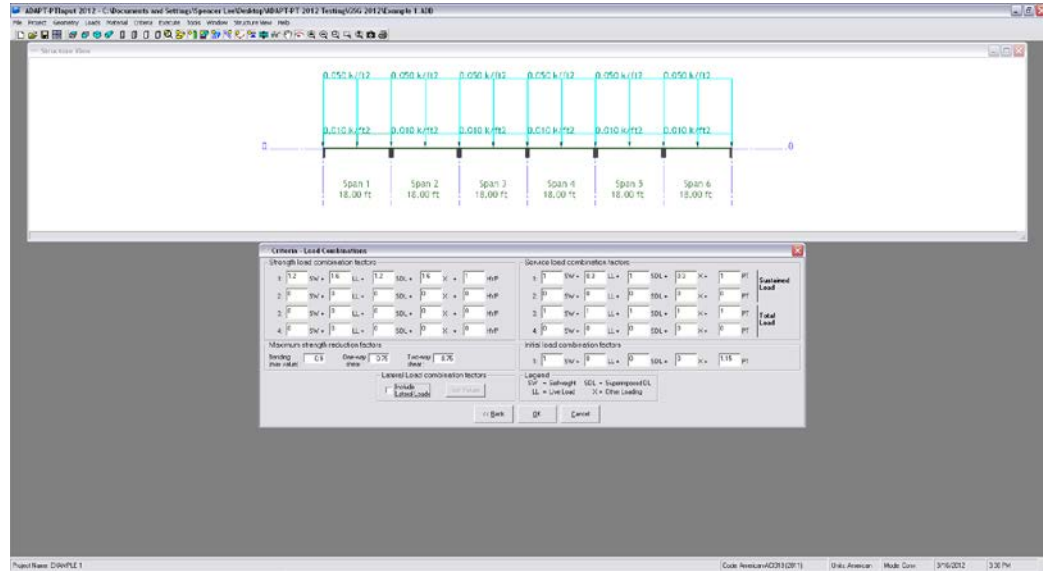



FIGURE 2.1-23

This is the last input screen. Click **OK** at the bottom of the screen to finish the data entry.

2.2 SAVE AND EXECUTE THE INPUT DATA

To save the input data and execute the analysis, either select **Execute Analysis** from the *Action* menu of the menu bar or click on the **Save & Execute Analysis** button . Then, give a **file name** and **directory** in which to save the file. The program now saves all sub-files in a specific folder with the name selected by the user, along with the .adb file at the user-defined directory. Once the file is saved, the program will automatically execute the analysis by reading the data files and performing a number of preliminary data checks.

Once the execution completes the selection of post-tensioning, the “PT Recycling” window, as shown in **Figure 2.2-1** opens. If an error is detected, the program will stop and display a message box indicating the most likely source of the error.

New!

The program now includes an option at the lower left corner of the “PT Recycling” window called “Typical Values Row.” When checked, the program includes a typical values input row in the “Current Tendon” and “All Tendons” input box. The user can enter typical values for “PT force” and “Tendon Control Point Height.” After entering the user-defined typical values, select ENTER on your keyboard and all data columns for each span will change to the typical value entered. See **Figure 2.2-1a**.

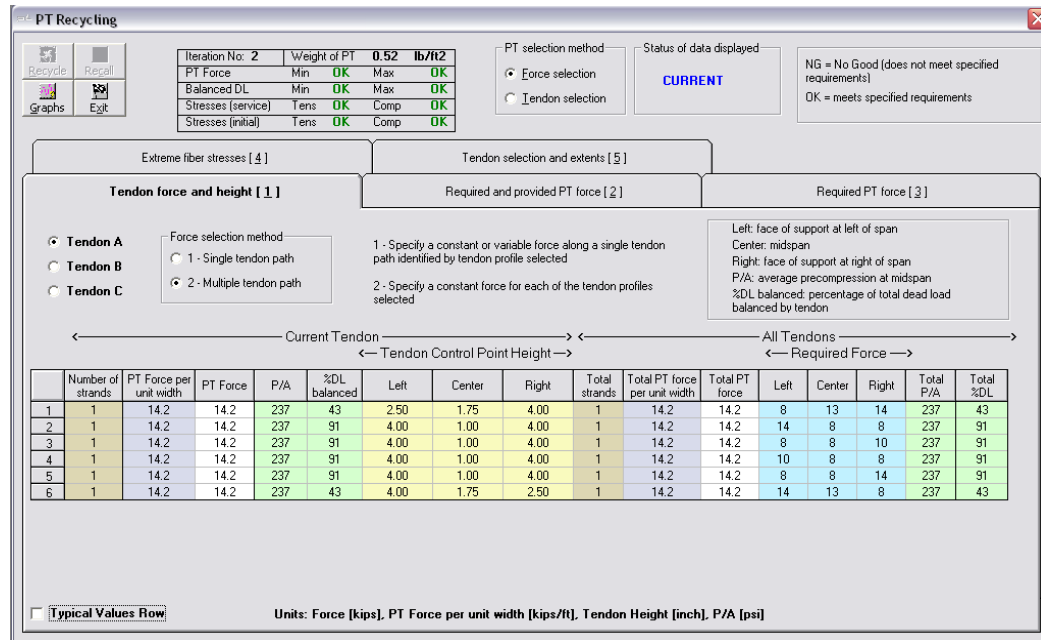


FIGURE 2.2-1

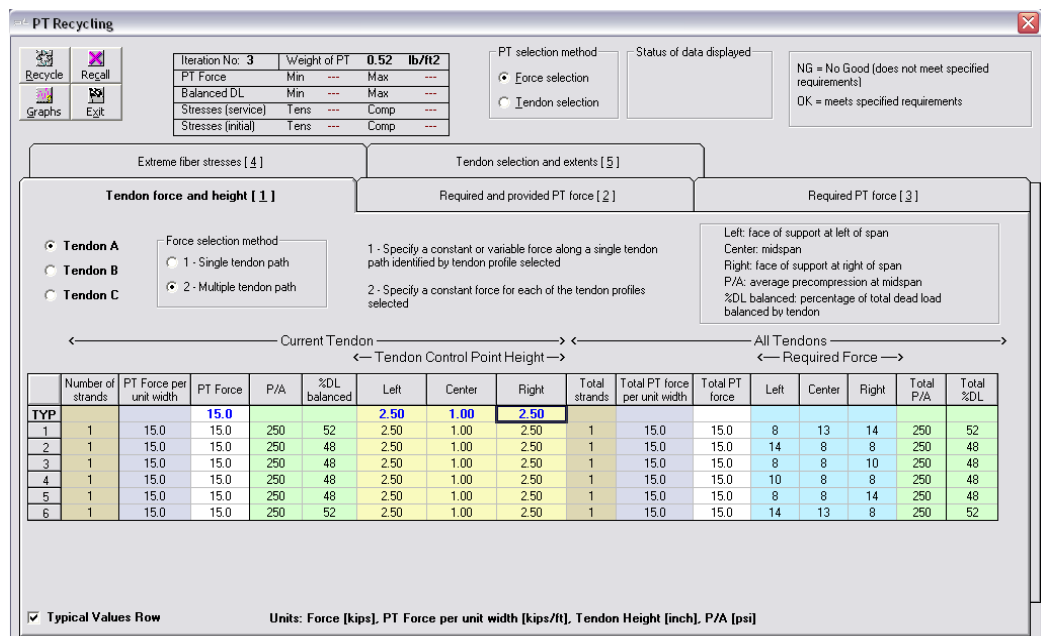
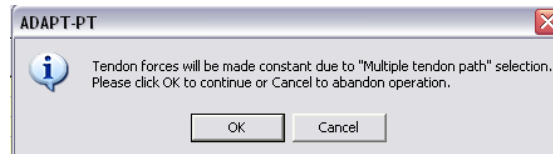


FIGURE 2.2-1a

New!

Here you can optimize the design by changing the tendon force and tendon heights. Turn off the “Typical Values Row” and select **Single tendon path** in the *Force selection method* so that you can specify a variable force along a single tendon. Note that if the option is changed back to **Multiple tendon path** the program will show the message below:



If the user selects “OK” the program will make the force constant and exit Recycler. If “Cancel” is selected the program will retain the current settings and remain in Recycler. For this example, **Single tendon path** will be used.

Change the first and last span force to **16 k/ft** (71.17 kN/m) and remaining spans to **10 k/ft** (40.03 kN/m). The values proposed by the program are acceptable, as indicated in the status box at the top of the display.

Once values are changed the status indicator at the top right of the Recycle window will begin to flash.

New!

Also, you can select the execution mode as either force selection or tendon selection. **Force selection** mode will be used for this example, but for explanation purposes, select **Tendon selection**. Note that when this option is selected, the **Single and Multiple tendon path** options become inactive and the Tendon selection and extents tab (No. 5) becomes active. Select this tab. The **Tendon Extents** graph now allows the user to shift the left and right ends of both Tendons B and C in the extents graph. The user can select the end of either tendon, select the CTRL key and shift the end. The table to the right of the graph has also been enhanced to allow the user to directly enter the **Left End** and **Right End** locations as a fraction of the span the ends are located in.

For example, Tendon B shown in **Figure 2.2-1b** is located at 1.0 at its left end and 2.5 at its right end. Tendon C is located at 2.5 at its left end and 4.0 at its right end. These values can be directly entered and the graphical view will update automatically or if the graphical view is updated, the tabular locations are updated.

Return back to the Single path mode by selecting **Force selection**. You will need to re-enter the force value in the interior spans to **10 k/ft** (40.03 kN/m).

Once all of the changes are made, as in **Figure 2.2-2**, click on the **Recycle** button to update all of the tabs, the “Design Indicator” box and the “Recycle Graphs”.

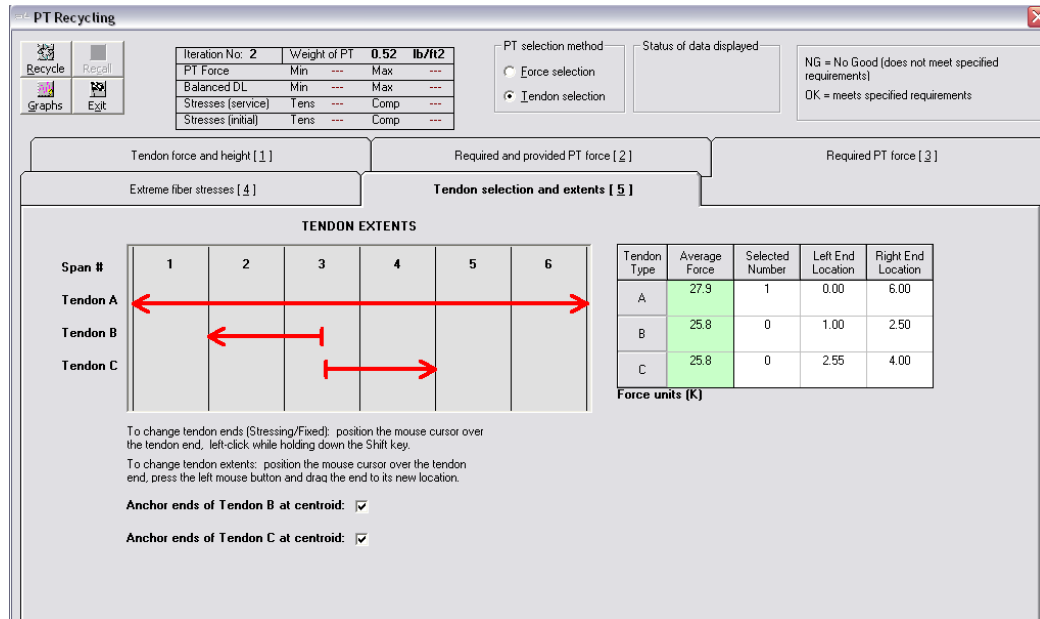


FIGURE 2.2-1b

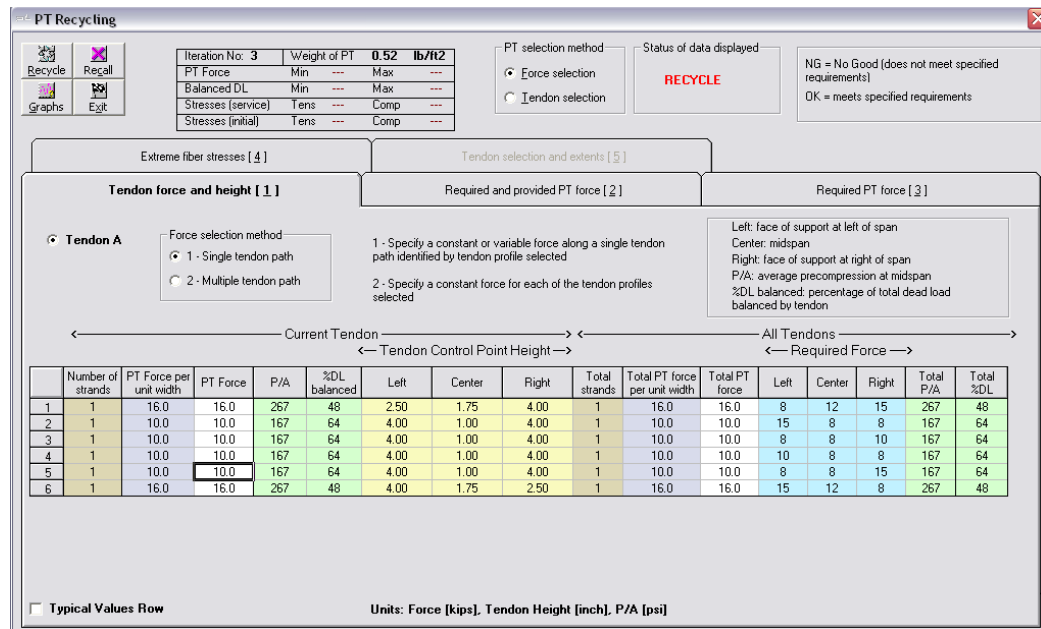


FIGURE 2.2-2

After the recalculation of the stresses and required forces along the member based on current values, the window, as shown in **Figure 2.2-3** with “Current” status indicator opens.

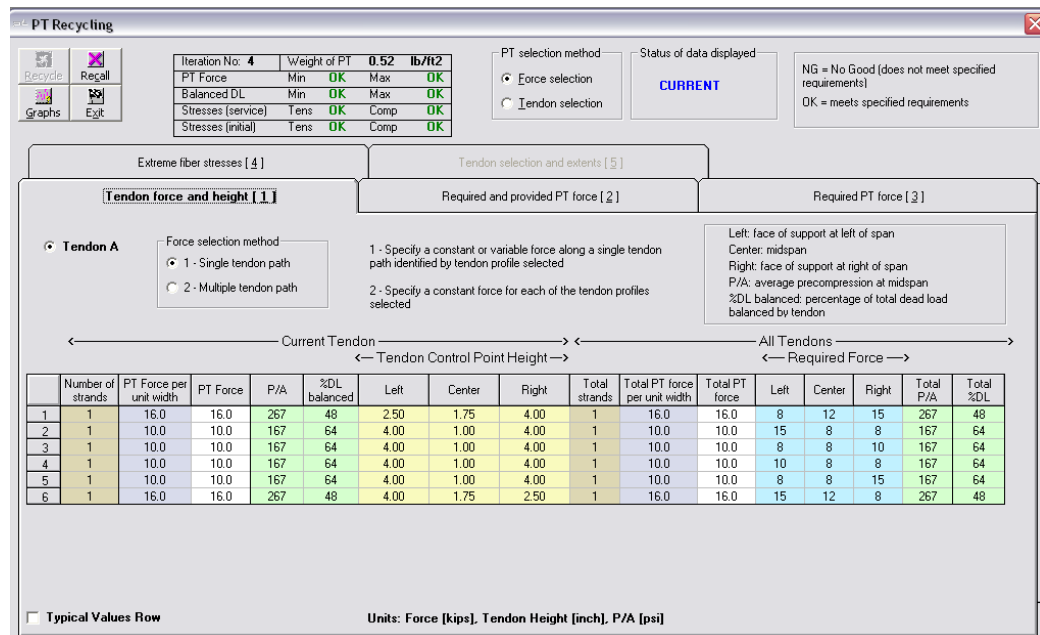


FIGURE 2.2-3

You can check the final stresses either by clicking **Extreme fiber stresses [4]** tab in the *PT Recycling* window (Fig. 2.2-3) or by clicking **Graphs** at the top left of the screen.

Graphs displays a set of three graphs which provide detailed information on the tendon profile, the tension and compression stresses and the required versus provided post-tensioning forces at 1/20th points along the spans (Fig. 2.2-4).

The top diagram, the **Tendon Height Diagram** shows the elevation of the tendon profile selected. The tendon profile can be viewed either with the concrete outline or without the concrete outline by checking the option at the left of the screen.

The second diagram, **Stress Diagrams**, plots the maximum compressive and tensile stresses at the top and bottom face of the member. You can view the stresses due to *Dead Load*, *Live Load*, *Post-tensioning* and *Service Combination* each separately, or in combination, by selecting the options on the screen. Also you can verify the top and bottom stresses due to service combination with the allowable values. In **Figure 2.2-4**, it shows the final bottom fiber stresses with the allowable values. In which, gray color represents the *allowable value*, *top curve* represents the bottom *tensile stress* and *bottom curve* represents the bottom *compressive stress*. If the calculated stress is not within the limit, i.e., the curve is outside the gray portion; you need to modify the forces to optimize the design. The third diagram, **Post-Tensioning Diagrams** shows the required and provided post-tensioning force at 1/20th points along each span. The *vertical line* represents the *required* post-tensioning and the *horizontal line* represents the *provided* post-

tensioning at that section. At each design section along a span, the program performs an analysis based on the post-tensioning force at that section.



FIGURE 2.2-4

If the solutions are not acceptable, you can change post-tensioning layout and, recycle until an acceptable solution is reached. Once you are satisfied with the solution, select **Exit** at the top left of the screen to continue with calculations.

The program continues with the calculations based on the most recent tendon forces and profile selections. Once finished successfully, you return to the main program window with the screen as shown in **Figure 2.2-5**.

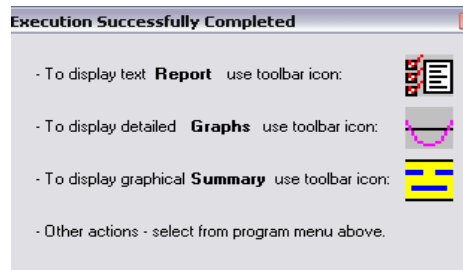



FIGURE 2.2-5

Close the above window by clicking **X** at the top right corner.

2.3 CREATE REPORTS

PT 2012 includes a Report Generator allowing the user to create full tabular, graphical reports or to customize any report according to predetermined report sections. To setup the report, select the **Report Setup** item on the *Options* menu or click the **Report Setup** button  on the main toolbar. The Report Generator screen shown in **Figure 2.3-1** will open.

The program allows you to generate reports in an MS-Word® editable format. You have the following options as explained below:

- Report cover: Select this option to generate a report cover with your logo and company information. To update your company information, click on **Update Company Info** on the *Report Generator* and you will see the screen **Company Information** shown in **Figure 2.3-2**.
- Table of Contents
- Concise Report: This report includes Project Design Parameters and Load Combinations as well as a Design Strip Report containing Geometry, Applied Loads, Design Moments, Tendon Profile, Stress check / Code check, Rebar Report, Punching Shear, Deflection and Quantities.
- Tabular Reports – Compact
- Tabular Reports – Detailed
- Graphical Reports
- Legend

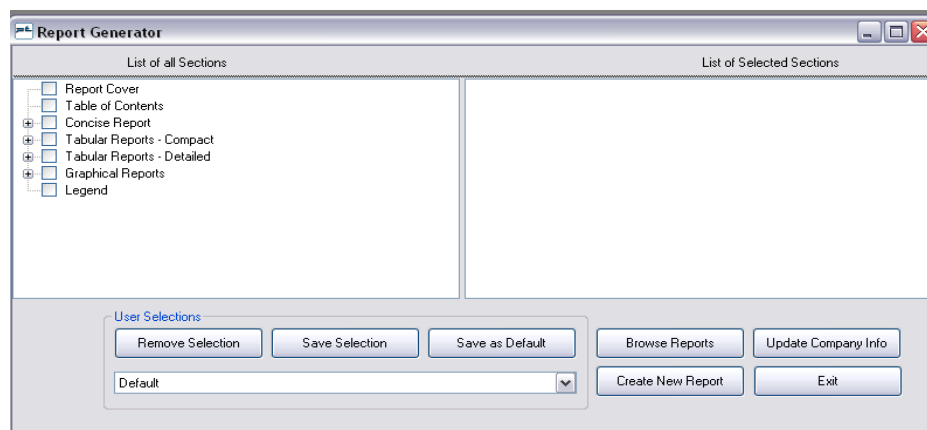


FIGURE 2.3-1

Simply check any item in the *List of all Sections* to include it in the report. The item will then appear in the *List of Selected Sections* on the right hand side of the *Report Generator*.

To generate and view the report, click on **Generate/View Report** on the bottom of the *Report Generator*.

The program allows you to open and view existing reports by clicking on **Open Reports**.

The Report Generator allows you to save report content as either a default template or as a user defined template. This enables you to quickly select content for any project by either using the default content or any other user defined content.

To define content as the default template, select the desired report content from the List of all Sections and click on **Save as Default**.

To define content as a user defined template, select the desired report content from the List of all Sections and click on Save Selection. You are asked to enter a name for your selection. This name will then appear in the drop down box in the **User Selections** frame.

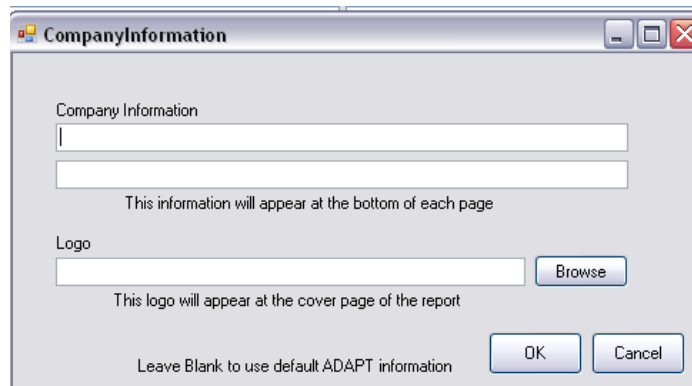




FIGURE 2.3-2

To open the “PT Summary Report” (**Fig. 2.3-3**) either click the **PTSum** button  on the tool bar or select the **PT summary** item on the *View* menu.

New to ADAPT-PT 2012 is the DXF option allowing the user to export a customizable, detailed drawing layout in .dxf format. The user can select options such as tendon profile, force data, profile values, system geometry and more. By Select the **PTSum** button and after selecting the DXF tab  the “Export Tendon Profiles to DXF Drawing” window will appear as shown is **Figure 2.3-4**. After making user-defined selections, press “Create DXF” to generate the .dxf file. See **Figure 2.3-5**.

To view the graphs, either click **Show Graphs** button  on the tool bar or select the **Graphs** item on the *View* menu.

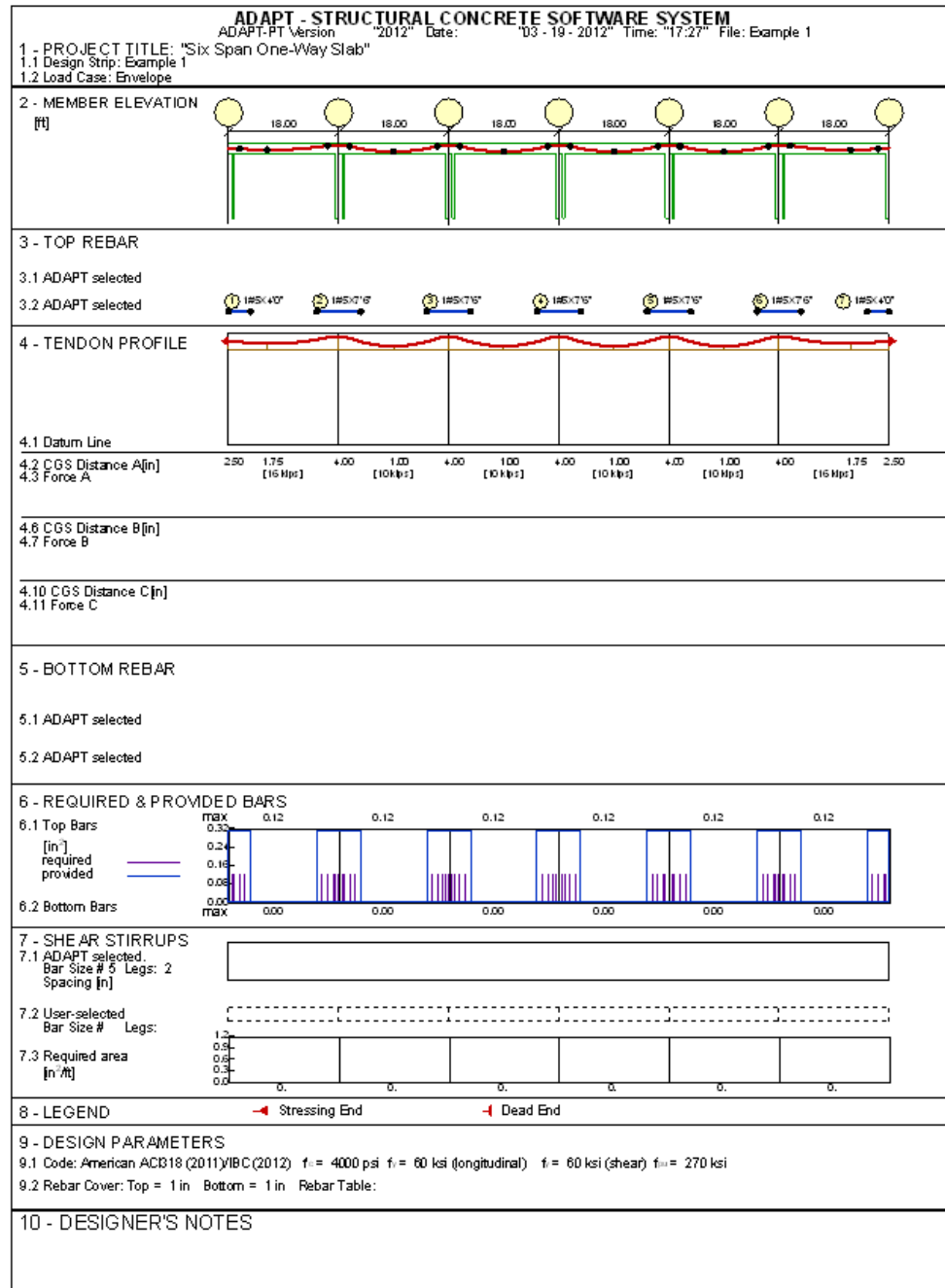


FIGURE 2.3-3

Export Tendon Profiles to DXF Drawing

Drawing Title
Six Span One-Way Slab - Example 1

Drawing Font
☒ Standard
☐ Arial
☐ Times New Roman

Font Scaling
 Large Font Scale: 1
 Small Font Scale: 1

Drawing Scaling
 Vertical Scale: 12
 Horizontal Scale: 1

Tendon Profile Settings
 Tendon Diameter [in]: 0.5
 Center Offset [in]: 0
 Height Roundup [in]: 0

Tendon Profiles Selection
☒ Tendon A
☐ Tendon B
☐ Tendon C

Tendon Height Reference Level
☒ Datum Line
☐ Structure Soffit

Tendon Height Location Level
☒ Tendon CGS Heights
☐ Tendon Support Heights

Tendon Drawing Arrangement
☒ Overlaid Tendon Profiles
☐ Stacked Tendon Profiles

Tendon Visualization Details
☒ Tendon Solid Profile
☒ Tendon Anchor Points
☒ Tendon Control Points

Tendon Height Details
☒ Tendon Heights Table
☒ Heights at Extreme Points
☐ Heights at 20th Points
☐ Heights at Interval Points
 0.0 [ft]
☐ Preset Support Heights

Change File... Create DXF

FIGURE 2.3-4

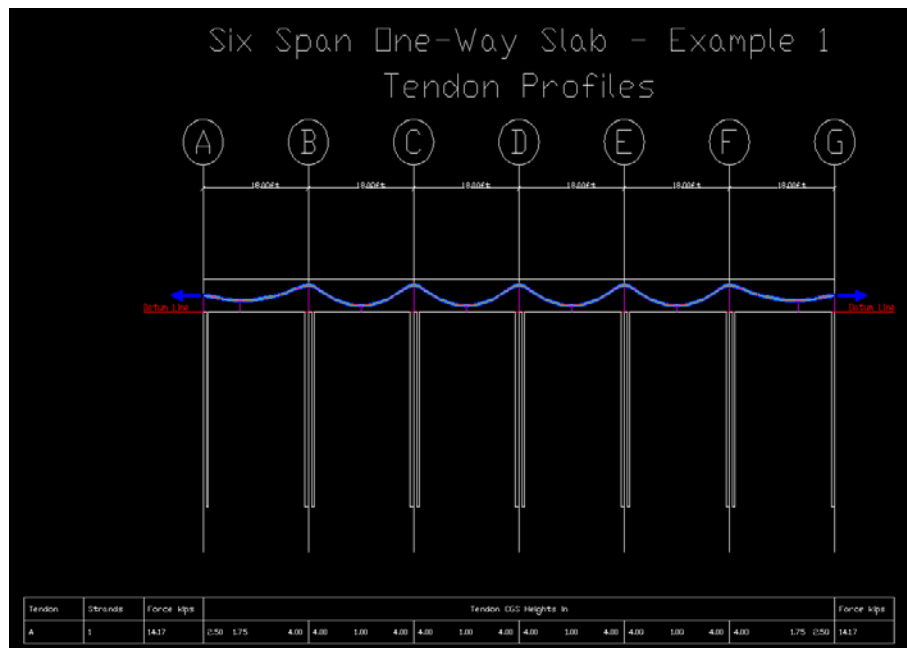


FIGURE 2.3-5

3 COLUMN-SUPPORTED SLAB (TWO-WAY SYSTEM)

Note that all new features described in the One-way Slab Example, unless noted otherwise, apply to two-way slab system models as well. Refer to Chapter 2 for new feature descriptions as they are not repeated in this Chapter.

The objective of this section is to demonstrate the step-by-step procedure in ADAPT-PT to generate data, analyze and design a column-supported slab. A column-supported slab is generally considered as a two-way system. This tutorial covers the following features of the program:

- Generation of input data, using the simple “Conventional” option of the program. Generation of data for complex geometry such as non-prismatic, segmented slabs, is not covered in this tutorial. Reference the ADAPT-PT 2012 User Manual for additional information.
- Design based on the “effective force” method, as opposed to selection of number of tendons. The application of the program for selection of number of tendons is not covered in this tutorial in its entirety. Reference the ADAPT-PT 2012 User Manual for additional information.

The structure selected is a typical design strip from a floor system. The geometry, material, loading and other particulars of the structure are given below. The geometry of the design strip of this tutorial is shown in **Figure 3-1**³.

Thickness of slab = 6.5 in (165.1 mm)

(i) Material Properties

o Concrete:

Compressive strength, f'_c	= 4000 psi	(27.58 MPa)
Weight	= 150 pcf	(2403 kg/m ³)
Modulus of Elasticity	= 3604 ksi	(24849 MPa)

o Prestressing:

Low Relaxation, Unbonded System

Strand Diameter	= ½ in	(13 mm)
Strand Area	= 0.153 in ²	(98 mm ²)
Modulus of Elasticity	= 28000 ksi	(193054 MPa)
Ultimate strength of strand, f_{pu}	= 270 ksi	(1862MPa)
Minimum strand cover		
From top fiber	= 1 in all spans	(25.4 mm)
From bottom fiber		

³ The geometry, loading, material properties and the design criteria selected are the same as those in PTI's publication for Design of Post-Tensioned Slabs.

Interior spans	= 1 in	(25.4 mm)
Exterior spans	= 1.5 in	(38.1 mm)

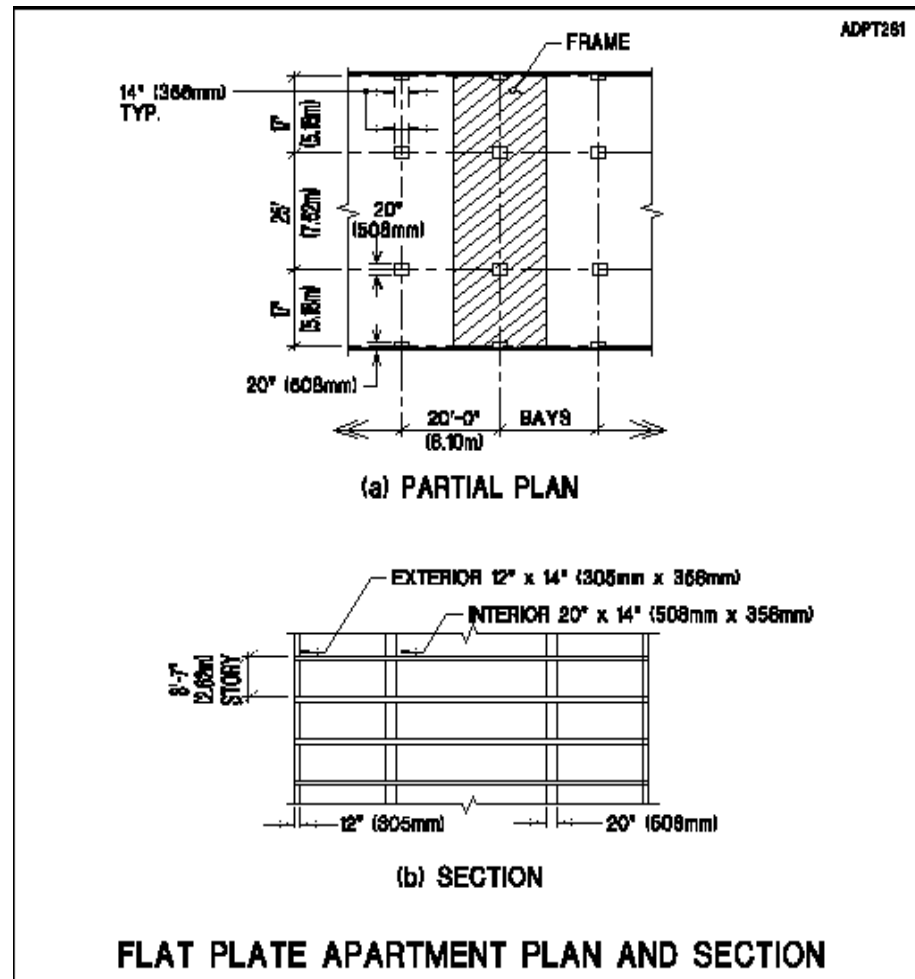


FIGURE 3-1

- Non-prestressed Reinforcement:

Yield stress, f_y	= 60 ksi	(413.69 MPa)
Modulus of Elasticity	= 29000 ksi	(199,949 MPa)
Minimum Rebar Cover	= 1in Top and Bottom	(25.4 mm)
- (ii) Loading**

Dead load	= self weight + 15 psf (SDL)
	= $(6.5/12) * 150 + 15$
	= 96 psf (4.60 kN/m ²)
Live load	= 40 psf (1.92 kN/m ²)

3.1 GENERATE THE STRUCTURAL MODEL

In the ADAPT-PT screen, click the *Options* menu and set the *Default Code* as **ACI318-2011/IBC 2012** and *Default Units* as **American**.

3.1.1 Edit the Project Information

3.1.1.1 General Settings (Fig. 3.1-1)

Open the new project by clicking either **New** on the *File* menu or the **New Project** button on the toolbar. This automatically opens the *General Settings* input screen, as in **Figure 3.1-1**. You can enter the *General Title* and /or *Specific Title* of the project in that window. For the purpose of this tutorial, enter the *General Title* as **Three-Span, Two-Way Slab**. This will appear at the top of the first page of the output. Enter *Specific Title* as **Example 2**. This will appear at the top of each subsequent page of the output.

Next, select *Geometry Input* as **Conventional**. Segmental input is used for entering non-prismatic structures, i.e., those where the tributary width or the depth of the section changes within a span.

Next, select the *Structural System* as **Two-Way slab**. Then there is an option to include drop caps, transverse beam and/or drop panels. In this case select **No**.

Click **Next** at the bottom right of this screen to open the next input screen, *Design Settings*.

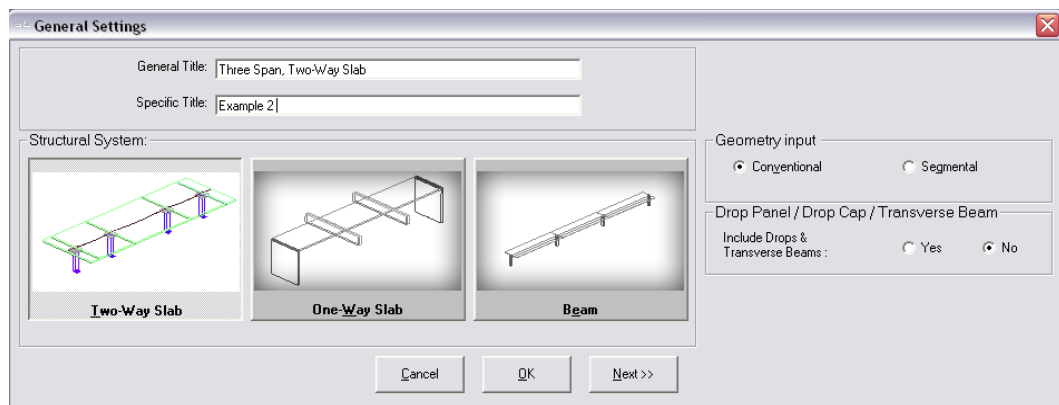


FIGURE 3.1-1

3.1.1.2 Design Code (Fig. 3.1-2)

The preferred design code will be specified in Step 2. In the Design Code screen, the code should be set as **American-ACI318(2011)/IBC 2012**.

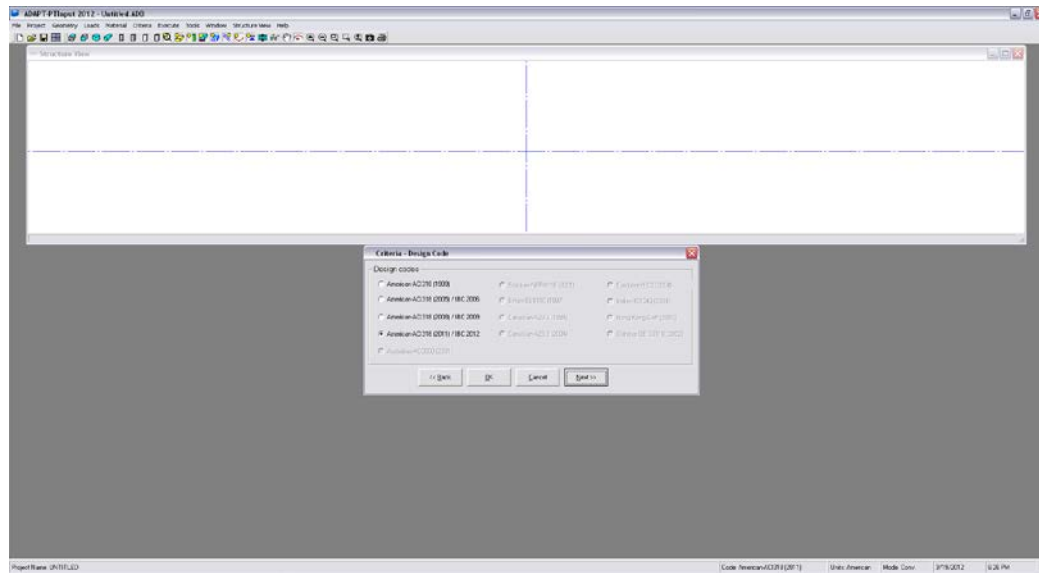


FIGURE 3.1-2

3.1.1.3 Design Settings (Fig. 3.1-3)

This screen is divided into four parts: *Analysis options*, *Design options*, *Contribution to unbalanced moment* and *Generate moment capacity based on*.

In *Analysis options*, you can select various calculation settings. First, select the *Execution Mode* as **Interactive**. In this mode, you have an opportunity to optimize the design by adjusting the tendon forces and tendon drapes for each span in the “Recycle” window. This will be explained later in this section.

Next, select **Yes** for *Reduce Moments to Face-of-Support* option. This invokes that the calculated centerline moments at each support are adjusted to the face-of support. In addition to the centerline moments, ADAPT-PT prints out the moments reduced to face-of- support. Select **No** for the option to *Redistribute moments*.

For two-way slab systems you have the option of modeling the structure either using the *Equivalent Frame method (EFM)* or *Simple Frame Method (SFM)*. Select **Yes** for the Equivalent Frame Modeling.

In *Design options*, you can either *Use all provisions of the code* that you have selected in the previous step, or *Disregard the following provisions* such as *Minimum rebar for serviceability*, *Design capacity exceeding cracking moment*, and *Contribution of prestressing in strength check*.

In *Contribution to unbalanced moment*, you either specify the contribution of *Top isolated bars*, and *Bottom isolated bars*, and *Post-tensioning* in percent. Leave the default values (100%).

Next is the option to *Generate moment capacity based on* **Design Values** or **User-Entered Values**. If **Design Values** is selected, the program will calculate and report positive and negative moment capacities based on prestressing steel, base reinforcement as defined by the user (this is discussed later in this section) and program-calculated reinforcement. Demand moments at 20th points along each span are also reported. When **User-Entered Values** is selected, the program will calculate and report similar moment capacities and demand moments where the capacities are based on prestressing steel and based reinforcement as defined by the user. For this example, select **Design Values**.

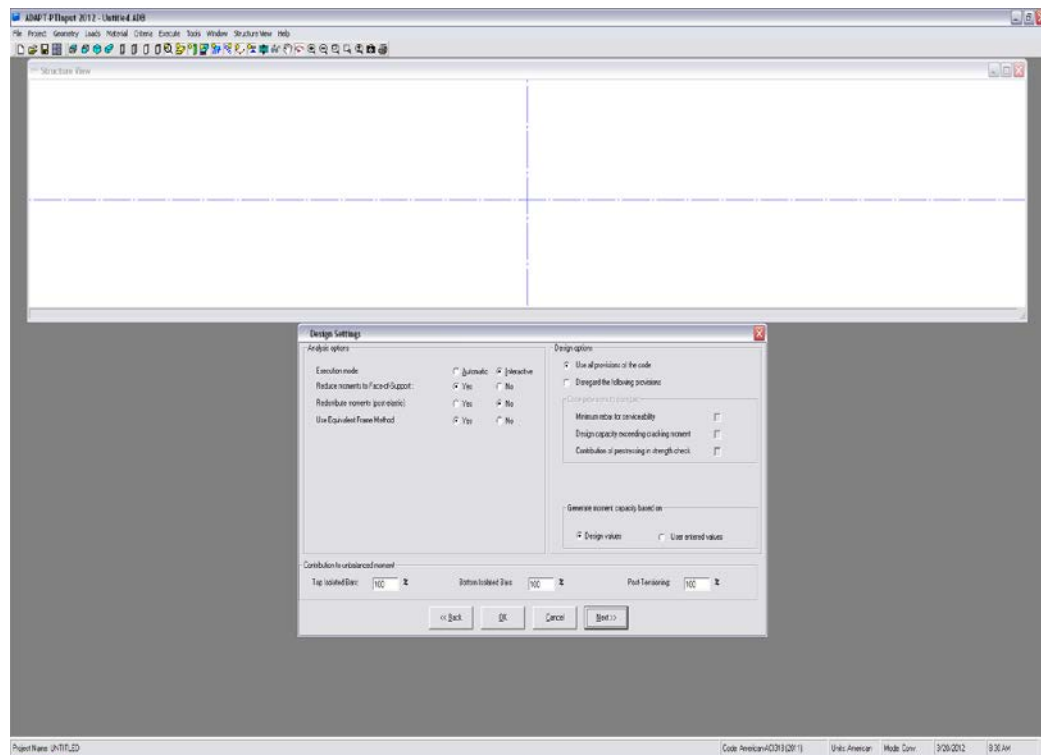


FIGURE 3.1-3

Click **Next** at the bottom right of the *Design Settings* screen to open the *Span Geometry* input screen.

3.1.2 Edit the Geometry of the Structure

3.1.2.1 Enter Span Geometry (Fig. 3.1-4)

This screen is used to enter the cross-sectional geometry of the slab at mid-span.

Set the *Number of Spans* as **3** either by clicking the **up arrow** or using **CTRL +**.

Next, enter the dimensions. All dimensions are defined in the legend at the top of the screen and/or illustrated in the appropriate section figure. The section type for any span can be changed by clicking on the button in the *Sec* (Section) column.

Select the section, *Sec*, as **Rectangular** and edit **17** ft (5.18 m) for length, *L*, **12** in (305 mm) for width, *b*, and **6.5** in (165 mm) for height, *h*, for all spans. You can use the “Typical” input row (top row) to enter similar dimensions. To enter typical values, type the **value** into the appropriate cell in the top row and then press **enter**. The typical value will be copied to all the spans.

As you enter the values, the span is displayed in real-time in the 3D window. You can zoom in and out in the *Structure View* with the help of your mouse wheel or with the help of the *Zoom In* or *Zoom Out* buttons in the *View Toolbar*.

You can access special data editing options by selecting data cells and right clicking. Available options include Insert New Line, Delete Line, Copy Selected Lines, and Paste Lines.

The reference height (*Rh*) identifies the position of a reference line that is used to specify the location of the tendon. Typically, the reference height is set equal to the slab depth. Click the **?** button with the **Rh** definition in the legend box to learn more about this feature. Edit reference height, *Rh* as **6.5** inches (165 mm), i.e., slab depth, for all spans.

The left and right multiplier columns (<-*M* and *M*->) are used to specify the tributary width to indicate how much of the tributary falls on either side of the support line. Tributary widths can be specified using either the Unit Strip method or the Tributary method. For this tutorial, unit strip (12 inches,(305 mm)) method is used, i.e., unit strip, **12** inches(305 mm) is entered as width, *b*, and the tributary width on either side of the support line is entered as the left and right multipliers.

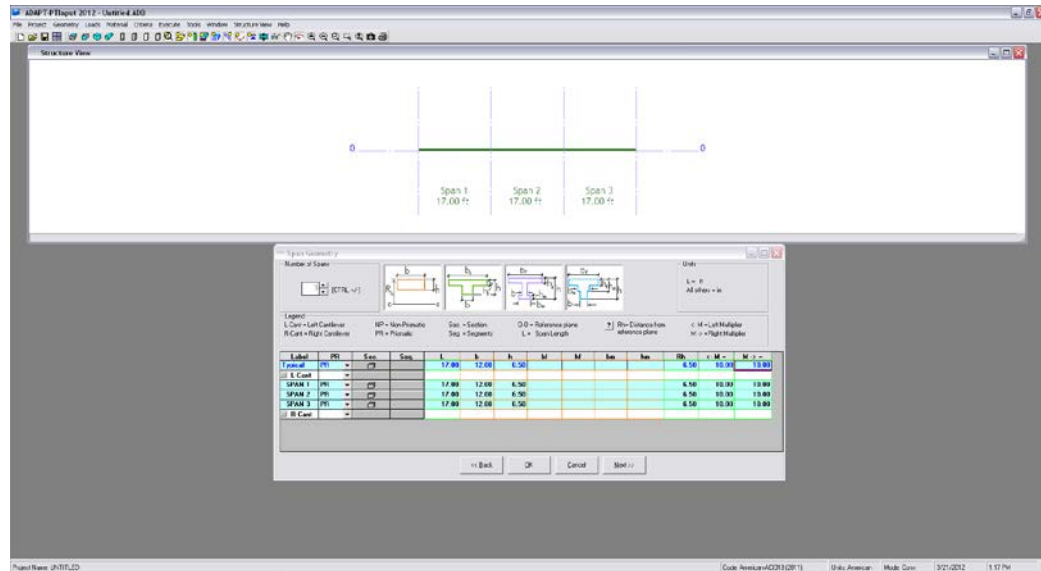


FIGURE 3.1-4

Click **Next** on the bottom line to open the next input screen.

3.1.2.2 Enter Support Geometry (Fig. 3.1-5)

This screen is used to input column /wall heights, widths and depths. You may enter dimensions for columns/walls above and/or below the slab.

Select the **Both Columns** from the support selection. Enter **8.58** ft (2.62 m) for **H1** and **H2** in the typical row and press **ENTER**, since all the supports are the same height.

Next, enter the dimensions of the supports. **B** is the dimension of the column cross- section normal to the direction of the frame. **D** is the column dimension parallel to the frame. Enter the given column dimensions as in **Figure 3.1-5**.

On this input screen, you can select for each support whether the left edge and the right edge of that support is interior or exterior. If the view of slab is shown as a Plan view, the left edge is that at the top of slab and the right edge is that at the bottom of slab. When **Exterior** is selected, the program automatically checks the distance from face of column to either edge. If this distance is less than 4*slab thickness, **h**, the program will consider the column an exterior condition (i.e. edge condition). If **Interior** is selected, the column will be treated as an interior condition.

For this example, all supports are **Interior** as the span is an interior span. Neither the left or the right span edge is an exterior edge.

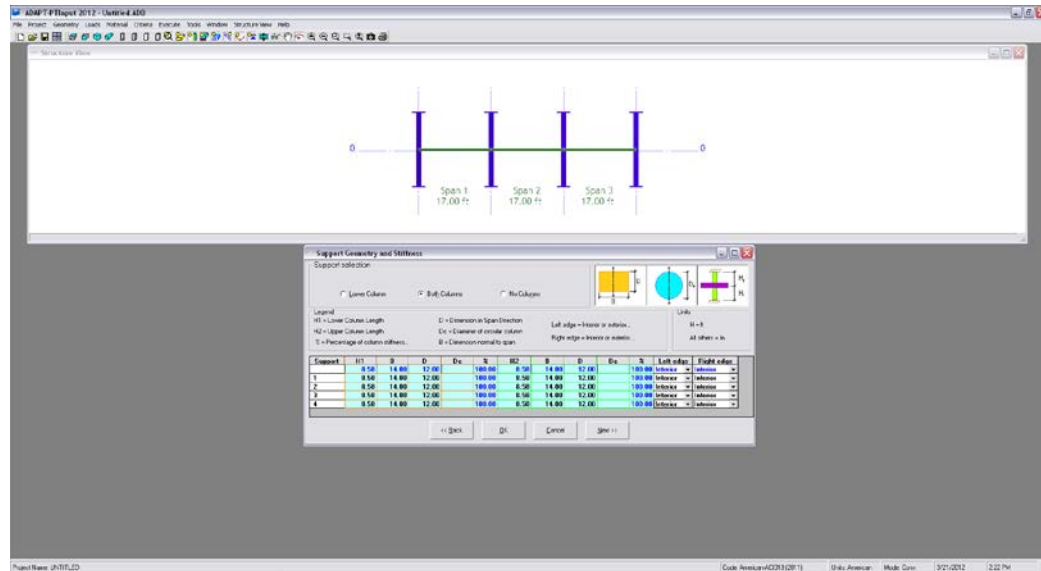


FIGURE 3.1-5

Click **Next** on the bottom line to open the *Supports Boundary Conditions* input screen.

3.1.2.3 Enter Support Boundary Conditions (Fig. 3.1-6)

This screen is used to enter support widths and column boundary conditions.

Support widths can be entered if you answered “Yes” to the “Reduce Moments to face-of- support” question on the *Design Settings* screen, i.e., if you answered “No”, you cannot input values in the SW column. This input value will be used to calculate the reduced moments.

Since the support width, *SW*, is set to the column dimension (*D*) as a default, the *SW* values will be automatically determined from the support geometry and cannot be modified by the user. If you want to input the *SW* values, **uncheck** the *SW=Column Dimension* box.

Select the boundary conditions for *lower* and *upper* columns as **1**(fixed) from the drop down list.

Leave the *End Support Fixity* for both the left and right supports as default **No**. This is used when the slab or beam is attached to an infinitely stiff member.

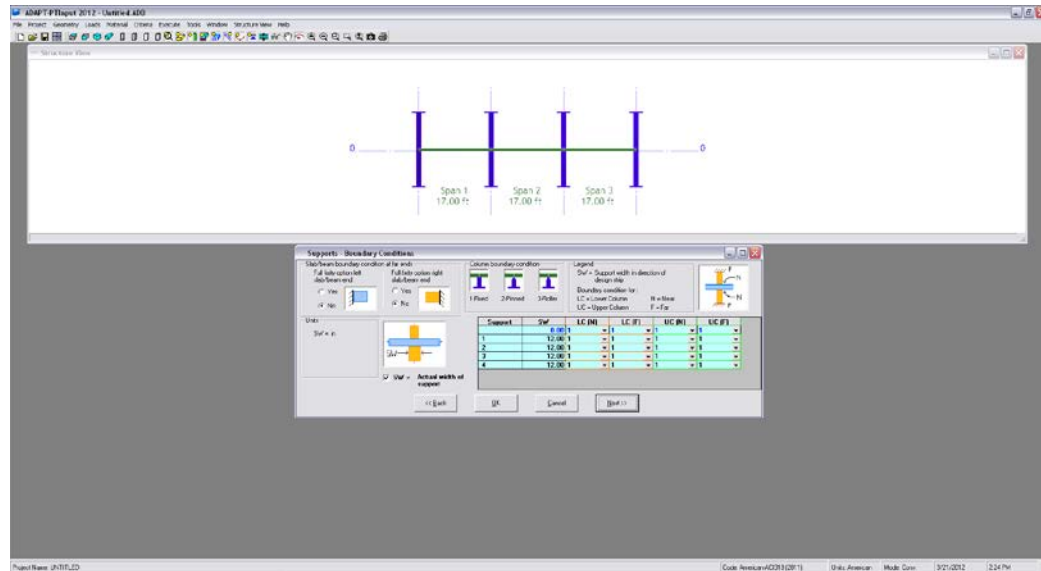


FIGURE 3.1-6

Click **Next** at the bottom of the screen to open the input screen *Loading*.

3.1.3 Enter Data

3.1.3.1 Edit the Loading Information (Fig. 3.1-7)

Any number of different loads and load types may be entered for a span.

Load types available in PT 2012 are: *Uniform, Partial Uniform, Concentrated, Moment (Concentrated), Line, Triangular, Variable and Trapezoidal.*

To enter loads specific to each span, enter the span number in the *Span* column. If the loads are the same for all the spans, you can type **ALL** or **all** in the *Span* column. This will copy the data to all of the spans. For this example, type all in the *Span* column.

If you choose not to include Self-weight, the self-weight (**SW**) can be input in its own load *Class*. In any case, you can choose to specify additional dead load as a superimposed dead load (**SDL**).

PT 2012 gives you the option to specify any other load (i.e. hydrostatic, soil, etc.) in the **X Class** loading.

Select the *Class* as **SDL** from the drop down list and specify the load type as uniform either by typing **U** in *L-?* or by **dragging the icon** from the graphics of the uniform loading. The default of the load type when you select the load class is L-U; so leave it as is for this tutorial.

Type **0.096 k/ft²** (4.60 kN/m²) for superimposed dead load in the *w* column. You can enter DL with or without self-weight, since the program can calculate self-weight automatically. In order to calculate the self-weight automatically, you must answer **Yes** to *Include Self-Weight* question at the top right of the screen and enter a **unit weight** of concrete.

Repeat the procedure for live load by entering **all** and changing the *Class* to **LL** and *w* value to **0.034 k/ft²** (1.63 kN/m²).

Answer **Yes** to *Skip Live Load?* at the top left of the screen and enter the *Skip Factor* as 1.

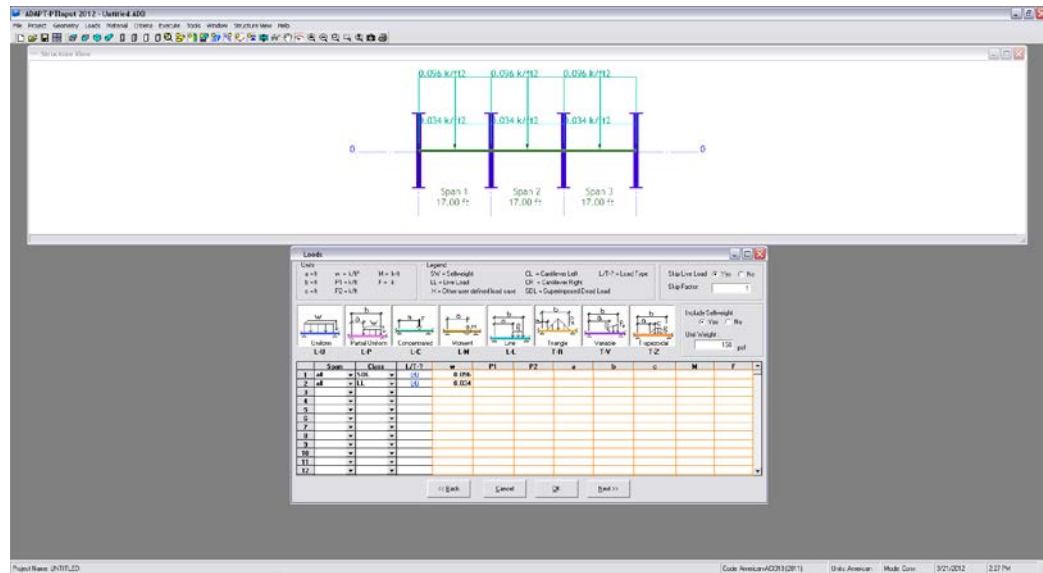


FIGURE 3.1-7

If you go to any other form and come back to the Loads input form, you will see that the loading information is now entered in the table for each span (**Fig. 3.1-8**).

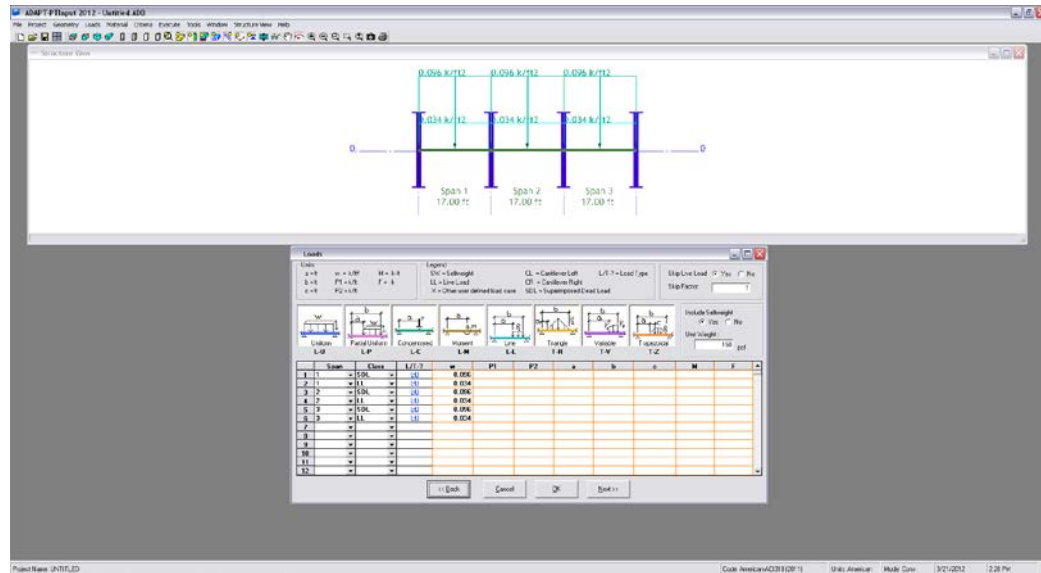


FIGURE 3.1-8

Click **Next** at the bottom of the screen to open the *Material - Concrete* input screen.

3.1.4 Edit the Material Properties

3.1.4.1 Enter The Properties Of Concrete (Fig. 3.1-9)

Select the **Normal** *weight* and enter the *strength at 28 days* for slab/beam and column to be **4000** psi. When you press **Enter** from the strength input value, the *Modulus of Elasticity* will be calculated automatically based on the concrete strength and the appropriate code formula.

For this tutorial, keep the default values of strength and creep coefficient. The creep coefficient will be used in the calculation of long-term deflection.

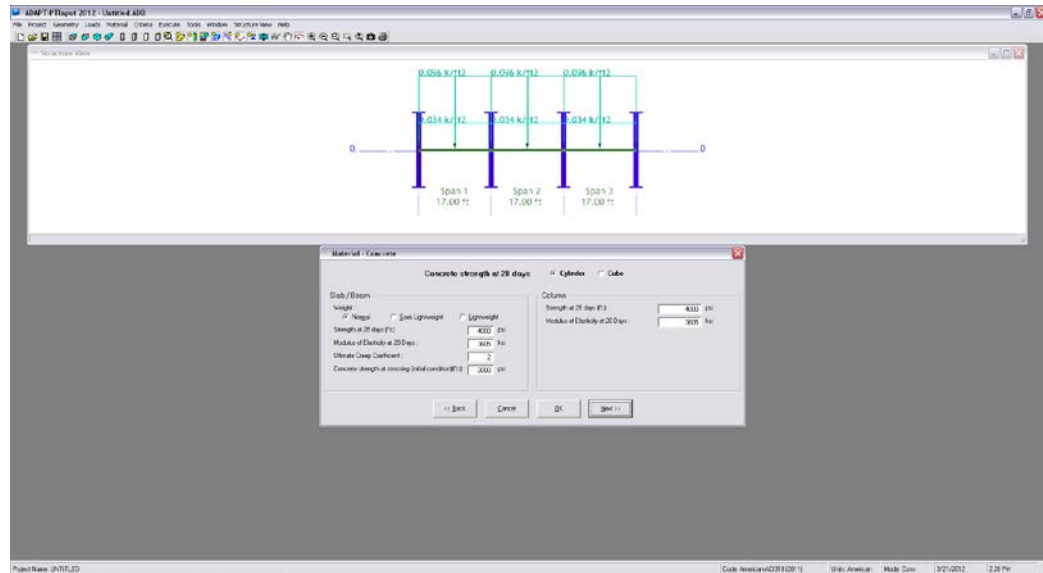


FIGURE 3.1-9

Click **Next** at the bottom of the screen to open the next input screen, *Material Reinforcement*.

3.1.4.2 Enter The Properties of Reinforcement (Fig. 3.1-10)

The screen is divided into two parts: *Longitudinal reinforcement* and *Shear reinforcement*.

In the section *Longitudinal reinforcement*, keep the default values for *Yield Strength* and *Modulus of Elasticity*. Change the *Preferred Bar Sizes for Top and Bottom* to **5** and **6** respectively (16, 19). These will be used when calculating the number of bars required.

In Shear reinforcement, select *Stirrup* and keep the default *Preferred Stirrup Bar Size* and the *Yield strength shear reinforcement*.

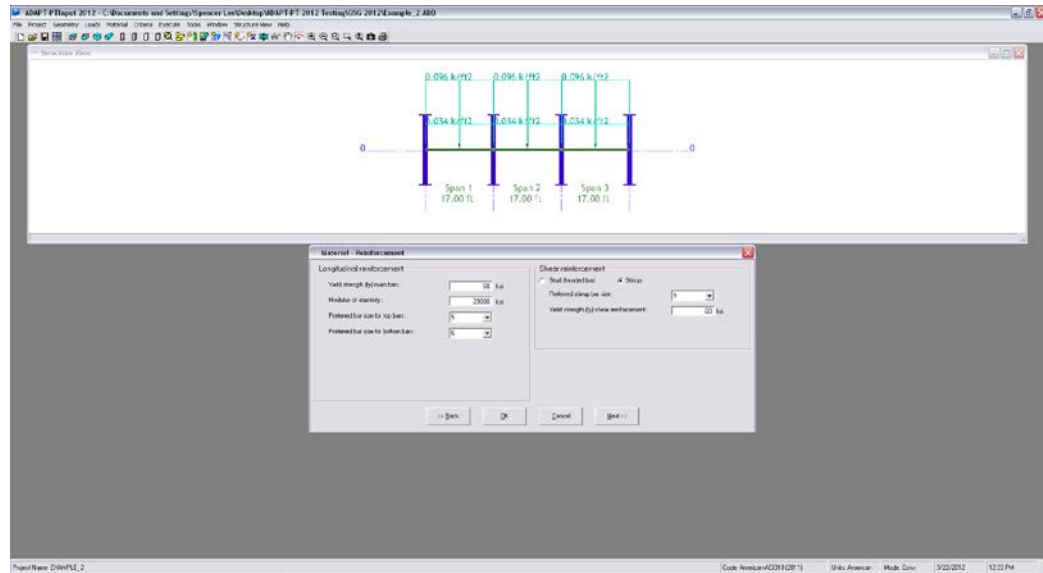


FIGURE 3.1-10

Click **Next** at the bottom of the screen to open the next input screen.

3.1.4.3 Enter The Post-Tensioning System Parameters (Fig. 3.1-11)

Select the *Post-tensioning system* as **Unbonded** and leave the default values of the other properties as they are.

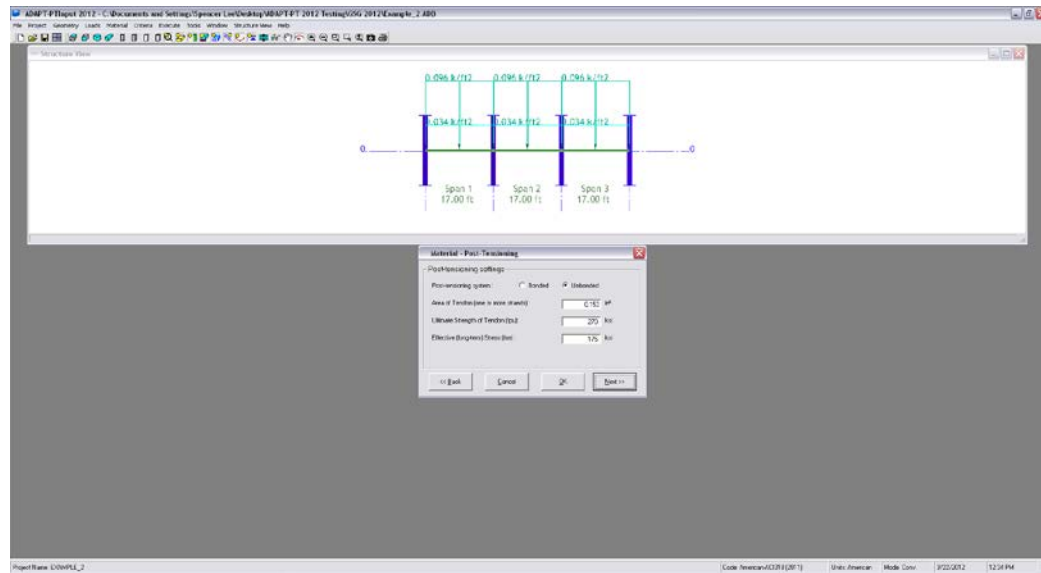


FIGURE 3.1-11

Click **Next** at the bottom of the screen to open the input screen, *Base Non-Prestressed Reinforcement*.

3.1.4.4 Edit Base Reinforcement (Fig. 3.1-12)

The program allows you to specify a base reinforcement that is taken into consideration when designing the structure. Select **Yes** in the *Base Reinforcement* section.

You have the choice between defining a mesh or isolated rebar. For this example choose **Isolated** from the drop down box.

Next specify the span where your base reinforcement starts. For this example, let the rebar start at the beginning of span 1. Therefore, enter a **1** in *First end location* and a **0** in $X1/L$.

If you wanted to let the rebar start mid-span of span 1, you could enter 0.5 for $X1/L$. For this example, leave the value as **0**.

To specify the end of the reinforcement at the end of span number 3, define **3** for *Second end location* and **1** for $X2/L$.

Furthermore, you specify **4** bars (*Number*) with *Bar Size* of **6** as **Bottom** bars with a *Cover* of **1** inch.

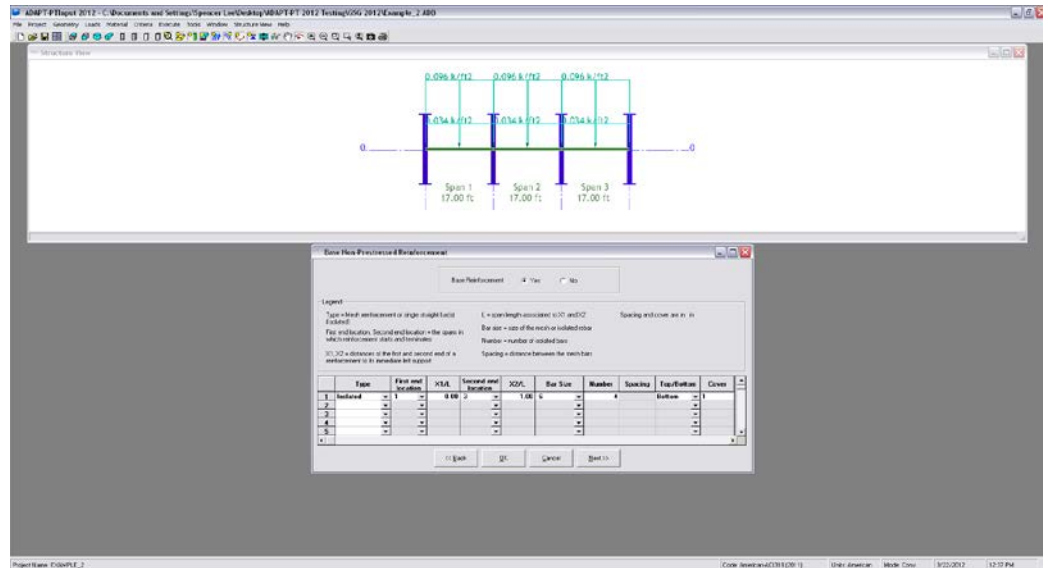


FIGURE 3.1-12

Click **Next** at the bottom of the screen to open the input screen, *Criteria – Allowable Stresses*.

3.1.5 Edit the design criteria

3.1.5.1 Enter The Initial And Final Allowable Stresses. (Fig. 3.1-13)

Tensile stresses are input as a multiple of the square root of f'_c , and compressive stresses are input as a multiple of f'_c .

The default values given in this screen are according to the applicable code. Allowable concrete stresses set forth in ACI 318-2011 are used for this example. Leave the default values as selected by the program.

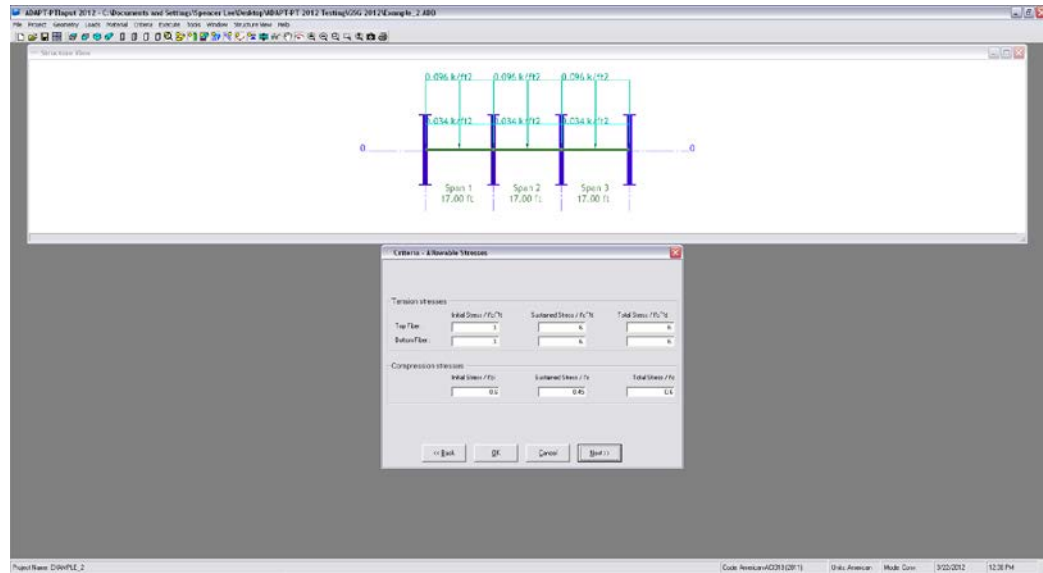


FIGURE 3.1-13

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Recommended Post-Tensioning Values*.

3.1.5.2 Enter The Recommended Post-Tensioning Values (Fig. 3.1-14)

This screen is used to specify minimum and maximum values for average precompression (P/A : total prestressing divided by gross cross-sectional area) and percentage of dead load to balance (W_{bal}). These values are used by the program to determine the post-tensioning requirements and the status of the P_{min}/P_{max} and $W_{BAL} Min/ Max$ indicators in the *Recycle window*.

The values given as default are according to code and the experience of economical design. Note that some codes may apply some lower and upper limit for average precompression (i.e. ACI 318-2011 sets the minimum allowable precompression to **125** psi for two-way slabs). Leave the default values as generated by the program.

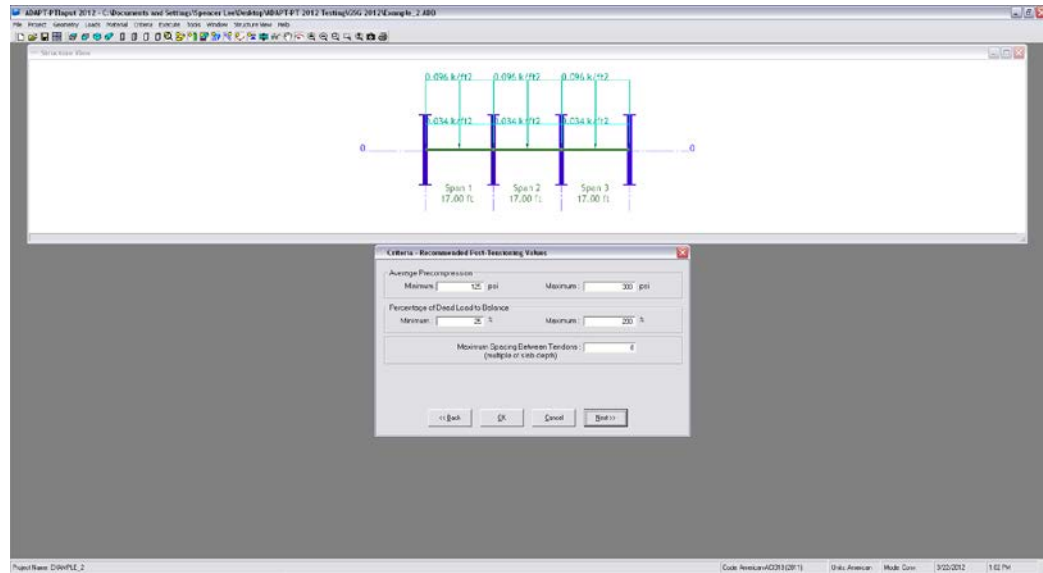


FIGURE 3.1-14

Click **Next** at the bottom of the screen to open the input screen, *Criteria – Calculation Options*.

3.1.5.3 Select The Post-Tensioning Design Option (Fig. 3.1-15)

The two design options are “Force selection” and “Calculate force/number of tendons,” as in **Figure 3.1-15**. **Force selection** is the default option and will be used for this example.

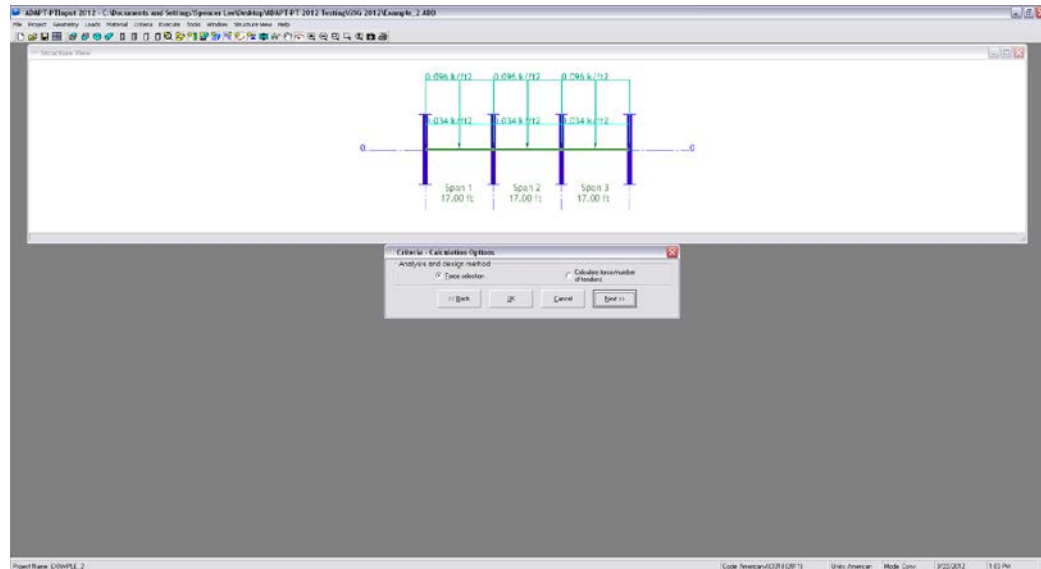


FIGURE 3.1-15

In this option, a tendon will be assigned a final and constant effective force, equal to the jacking force minus all stress losses, expressed as a single value.

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Tendon Profile*.

3.1.5.4 Specify The Tendon Profiles (Fig. 3.1-16)

The program allows you to specify up to three tendon paths per span. You can define one profile for each of the three tendons.

In the section *Option for tendons* you can define the *Default extension of terminated tendon as fraction of span*. Also, you can specify the *Shape of tendon extension* from the *Left end* and the *Right end*. For this example, leave the default values.

For this example, only Tendon A will be used. From the *Type* drop down list, select **1** for the reversed parabola option and change the inflection points ($X1/L$ & $X3/L$) to **zero**, since we assume a simple parabola with no inflection points. For the second span, keep the low point ($X2/L$) at mid span, i.e., at **0.5**. From the calculation, the low point for the first and third spans are at $0.490 \cdot L$ and $0.510 \cdot L$,

respectively from the left support. Enter $X2/L$ for the first and third spans as **0.490** and **0.510**, respectively.

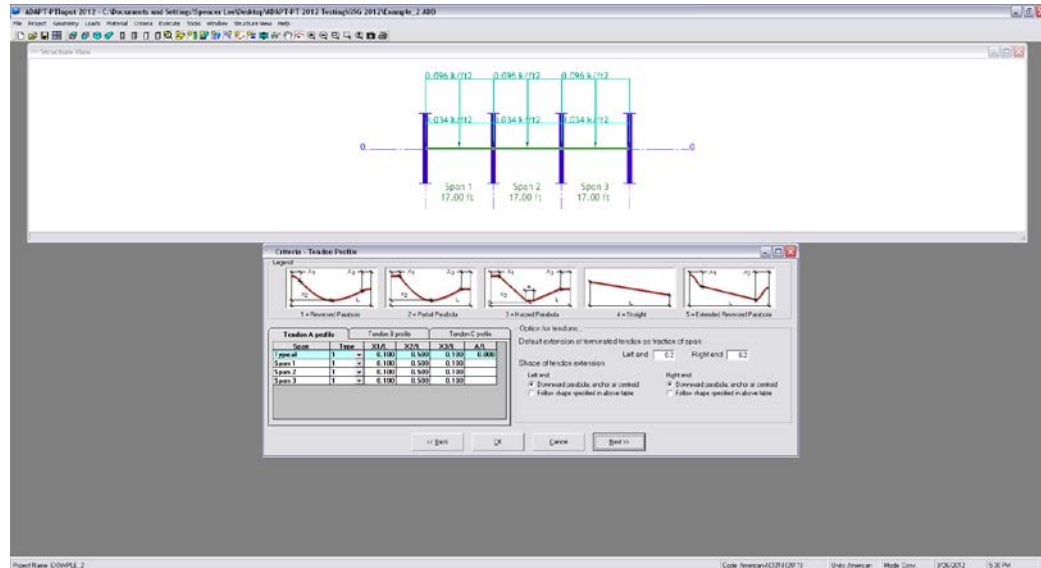


FIGURE 3.1-16

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Minimum Covers*.

3.1.5.5 Specify Minimum Covers For Post-Tensioning Tendons And Mild Steel Reinforcement (Fig. 3.1-17)

The cover for the prestressing steel is specified to the center of gravity of the strand (cgs). Therefore, for ½ inch diameter (13 mm) strands, the cgs is minimum cover + ½ * ½, i.e., cgs = cover + 0.25" (cgs = cover + ½ * 13). Edit CGS of the tendon as **1.25** inches (32 mm) for both the top fiber and the interior spans of bottom fiber and **1.75** inches (44 mm) for the exterior spans for the bottom fiber.

For non-prestressed reinforcement, edit **1 in** (25 mm) *Cover* for both the top and the bottom.

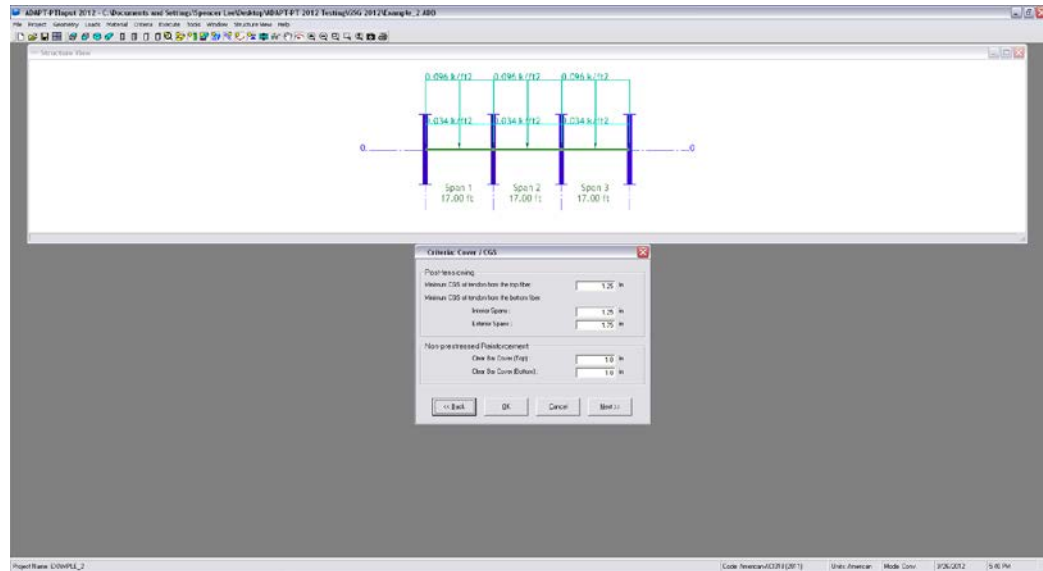


FIGURE 3.1-17

Click **Next** at the bottom of the screen to open the input screen, *Criteria - Minimum Bar Extension*.

3.1.5.6 Specify Minimum Bar Length and Bar Extension of Mild Steel Reinforcement (Fig. 3.1-18)

The values given as default are according to the appropriate code, for this tutorial, the ACI 318-2011 code. Leave the default values for this example.

The values entered for cut-off lengths are used to calculate top and bottom bar lengths when minimum reinforcement requirements govern.

The development length of reinforcement required for strength will extend the reinforcement by the given value beyond the calculated length. Please note that the program does not calculate this value automatically per bar size used.

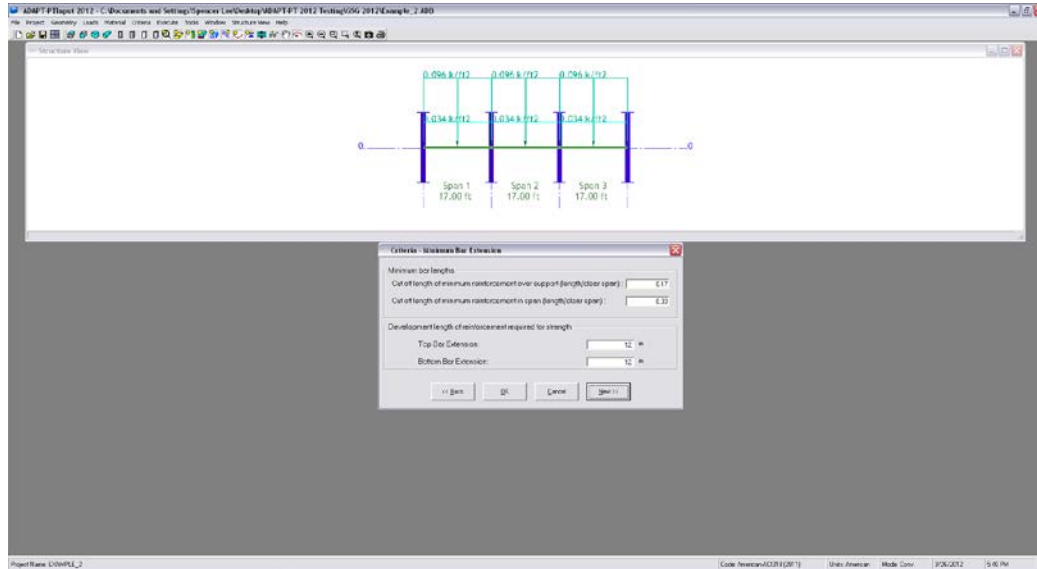


FIGURE 3.1-18

Click **Next** at the bottom of the screen to open the input screen, *Load Combinations*.

3.1.5.7 Input Load Combinations (Fig. 3.1-19, 20, 21)

Figure 3.1-19 shows the screen which is used to input the load combination factors for service and strength (ultimate) load conditions. It is also used to enter any applicable strength reduction factors. The default values are according to the ACI 318-2011 code. Leave the default values for this example.

The program allows you to specify four strength load combinations and four service load combinations. For ACI 318-2011, two of the service load combinations are reserved for sustained load and two for total load.

To include lateral loads, check the box to *Include lateral loads* and click on the *Set Values* button to define *Lateral moments* (**Fig. 3.1-20**) and *Lateral load combinations* (**Fig. 3.1-21**). For this example, lateral loads will not be considered. See Page 30 for a description of new default load combinations in ADAPT-PT 2011.

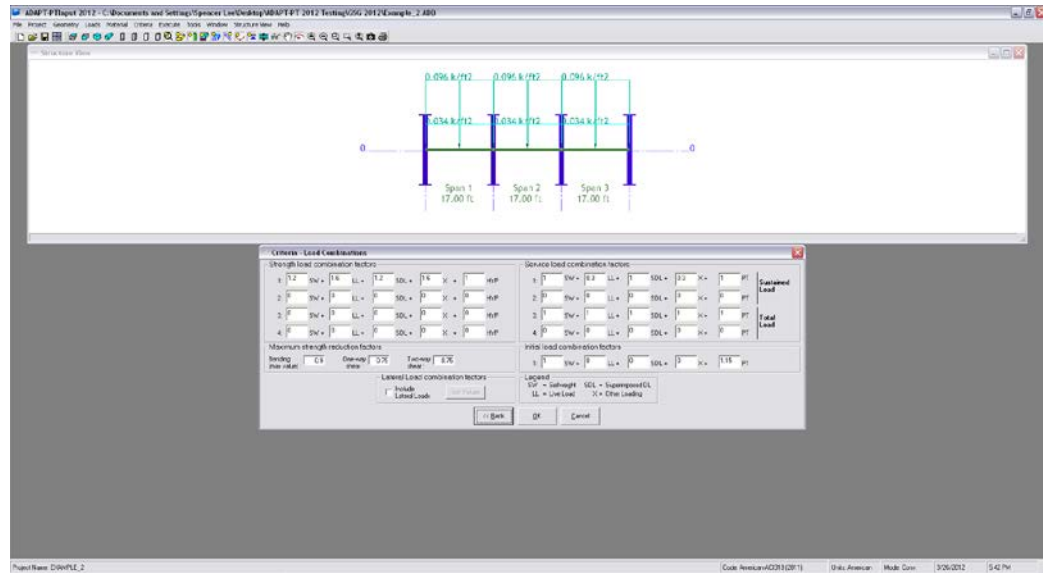


FIGURE 3.1-19

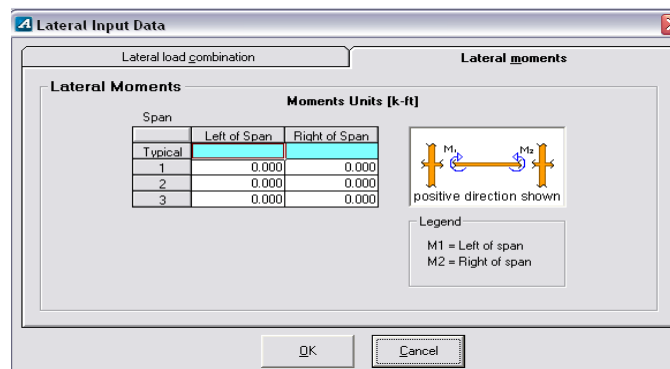


FIGURE 3.1-20

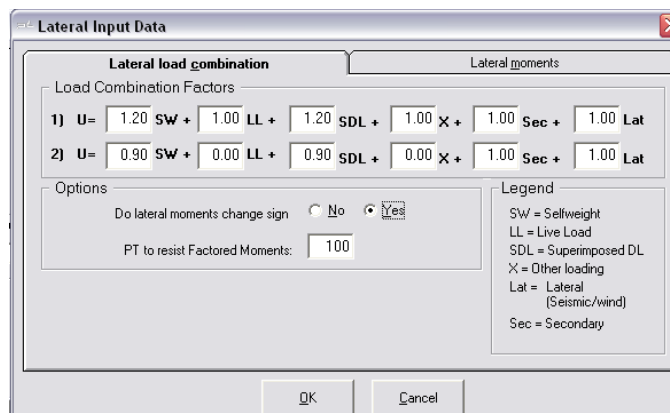



FIGURE 3.1-21

Click **OK** at the bottom of the screen to finish the input wizard.

3.2 SAVE AND EXECUTE THE INPUT DATA

To save the input data and execute the analysis, either select **Execute Analysis** from the *Action* menu of the menu bar or click on the **Save & Execute Analysis** button . Then, give a **file name** and **directory** in which to save the file. The program saves all sub-files in a specific folder with the name selected by the user, along with the .adb file at the user-defined directory. Once the file is saved, the program will automatically execute the analysis by reading the data files and performing a number of preliminary data checks.

Once the execution completes the selection of post-tensioning, the *PT Recycling* window, as shown in **Figure 3.2-1** opens. If an error is detected, the program will stop and display a message box indicating the most likely source of the error.

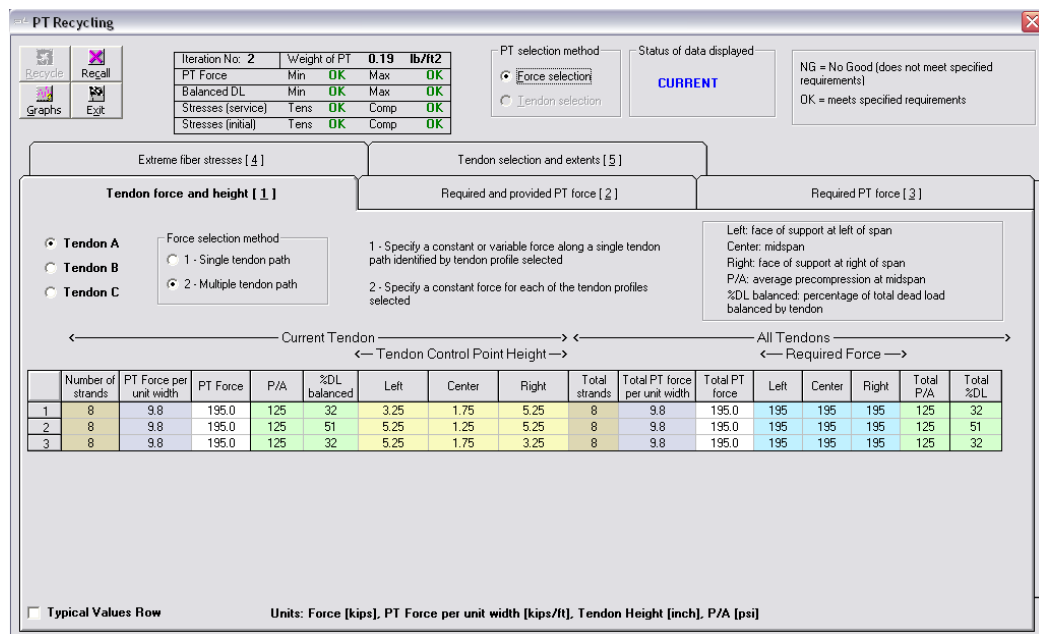


FIGURE 3.2-1

Here you can optimize the design by changing the tendon forces and tendon heights. Select **1-Single tendon path** for the *Force selection method*. Change the first and third span force to **206.5 k/ft (918.55 kN/m)** and the second span to **201.5 k/ft (896 kN/m)**. The status indicator at the top right of the Recycle window will begin to flash.

Since we selected the “Force Selection” option during data entry, the program will only allow the “Force Selection” mode for execution.

Once all of the changes are made as shown in **Figure 3.2-2**, click on the **Recycle** button to update all of the tabs, the Design Indicator box and the Recycle Graphs.

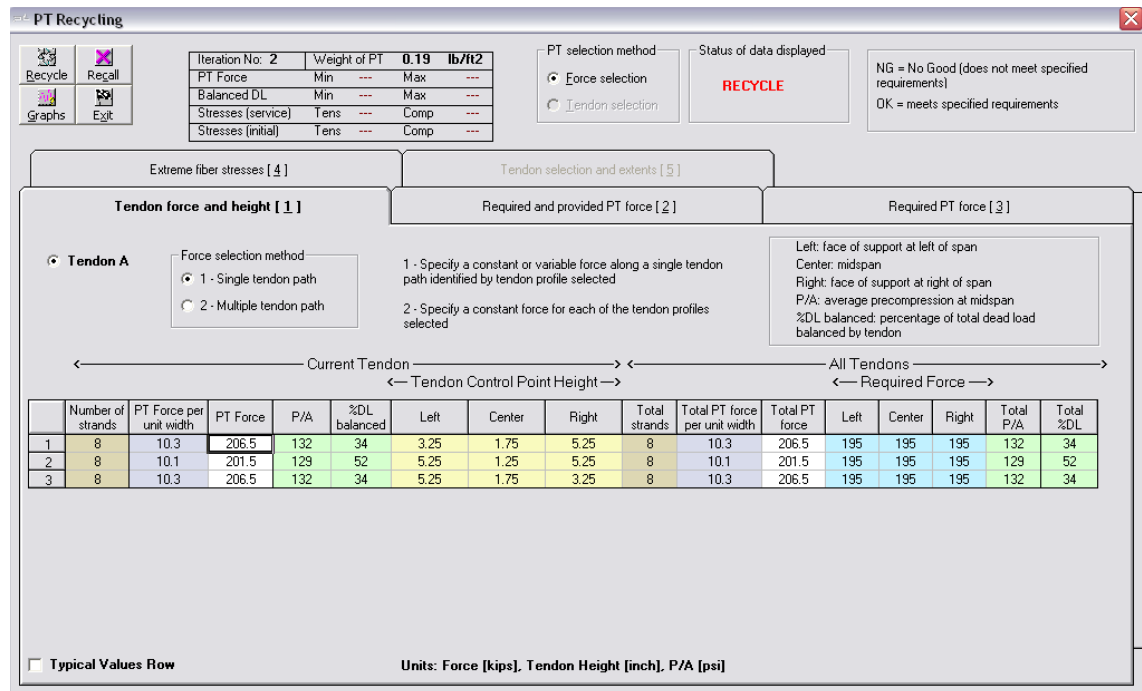


FIGURE 3.2-2

After the recalculation of the stresses and required forces along the member, based on the current values, the window, as shown in **Figure 3.2-3**, with the “OK” status for all items in the *design indicator box* opens.

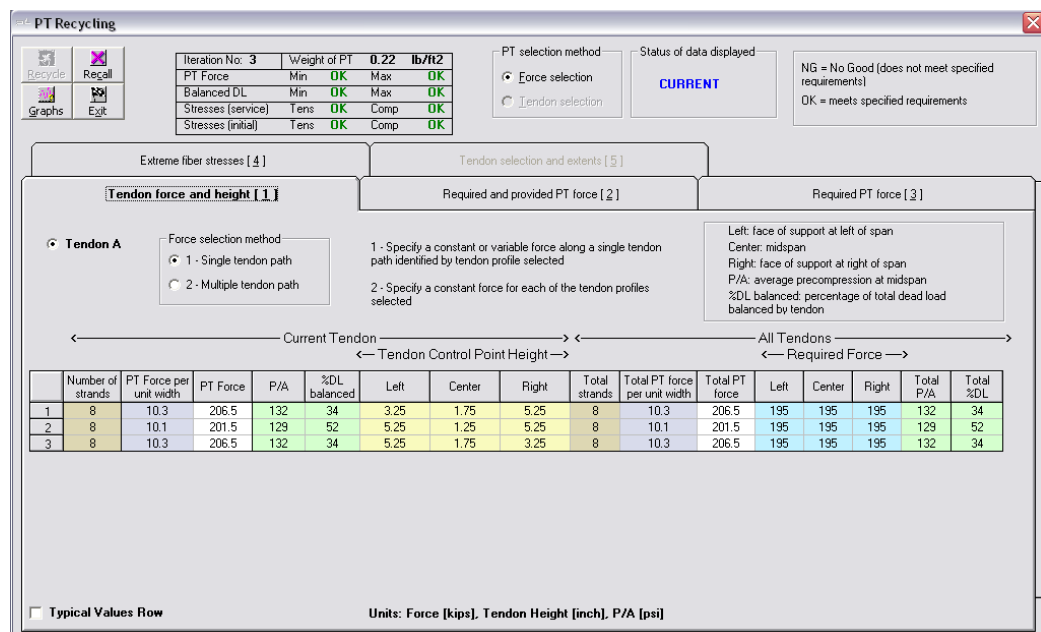


FIGURE 3.2-3

You can check the final stresses either by clicking **Extreme fiber stresses [4]** tab in the *PT Recycling* window (**Fig. 3.2-3**) or by clicking **Graphs** at the top left of the screen.

Graphs displays a set of three graphs which provide detailed information on the tendon profile, the tension and compression stresses and the required versus provided post-tensioning forces at 1/20th points along the spans (**Fig. 3.2-4**).

The top diagram, the **Tendon Height Diagram** shows the elevation of tendon profile selected. Tendon profile can be viewed either with concrete outline or without concrete outline by checking the option at the left of the screen.

The second diagram, **Stress Diagrams**, plots the maximum compressive and tensile stresses at the top and bottom face of the member. You can view the stresses due to e.g. *Self weight*, *Superimposed Dead Load*, *Live Load*, *Post-tensioning* and *Sustained* each separately, or in combination, by selecting the options at the screen. Also you can verify the top and bottom stresses due to the service combination with the allowable values. In **Figure 3.2-4**, it shows the final top fiber stresses with the allowable stresses. In which, *gray color* represents the allowable value, *top curve* represents the tensile stress and *bottom curve* represents the compressive stress. If the calculated stress is not within the limit, i.e., the top or bottom curve is outside the gray portion; you need to modify the forces to optimize the design.

The third diagram, **Post-Tensioning Diagrams** shows the required and provided post-tensioning force at 1/20th points along each span. The *vertical line* represents the *required* post-tensioning and the *horizontal line* represents the *provided* post-tensioning at that section. At each design section along a span, the program performs an analysis based on the post-tensioning force at that section.

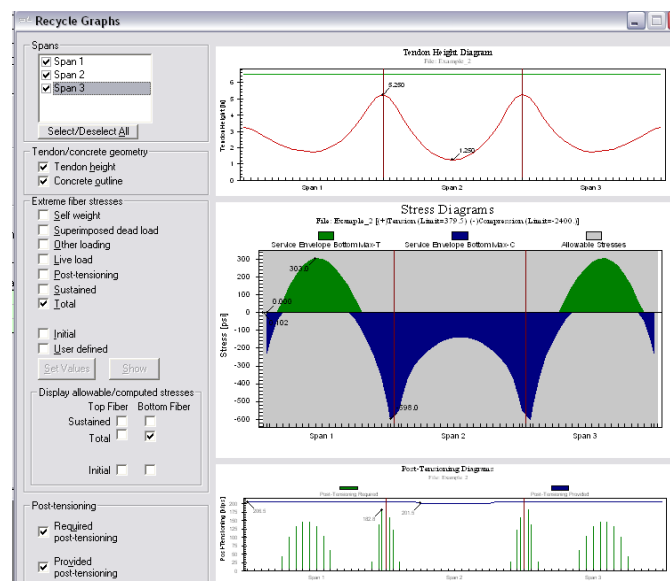


FIGURE 3.2-4

If the solutions are not acceptable, you can change post-tensioning layout and recycle until an acceptable solution is reached. Once you are satisfied with the solution, select **Exit** at the top left of the *PT Recycling* screen to continue with the calculations.


The program continues with the calculations based on the most recent tendon forces and profile selection. Once successfully finished, you return to the main program window with the screen as shown in **Figure 3.2-5**.



FIGURE 3.2-5

Close the above window by clicking **X** at the top right corner.

3.3 CREATE REPORTS

PT 2012 includes a Report Generator allowing the user to create full tabular, graphical reports or to customize any report according to predetermined report sections. To setup the report, select the **Report Setup** item on the *Options* menu or click the **Report Setup** button  on the main toolbar. The Report Generator screen shown in **Figure 3.3-1** will open.

The program allows you to generate reports in an MS-Word® editable format. You have the following options as explained below:

- Report cover: Select this option to generate a report cover with your logo and company information. To update your company information, click on **Update Company Info** on the *Report Generator* and you will see the screen **Company Information** shown in **Figure 3.3-2**.
- Table of Contents
- Concise Report: This report includes Project Design Parameters and Load Combinations as well as a Design Strip Report containing Geometry, Applied Loads, Design Moments, Tendon Profile, Stress check / Code check, Rebar Report, Punching Shear and Deflections. The program now reports Material Quantities in this report.
- Tabular Reports – Compact
- Tabular Reports – Detailed: This report now includes Demand Moments and Moment Capacities
- Graphical Reports

- Legend

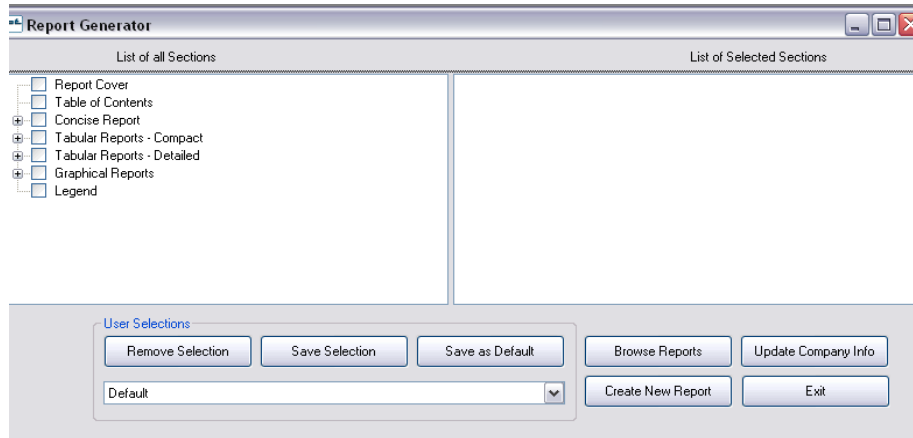


FIGURE 3.3-1

Simply check any item in the *List of all Sections* to include it in the report. The item will then appear in the *List of Selected Sections* on the right hand side of the *Report Generator*.

To generate and view the report, click on **Generate/View Report** on the bottom of the *Report Generator*.

The program allows you to open and view existing reports by clicking on **Open Reports**.

The Report Generator allows you to save report content as either a default template or as a user defined template. This enables you to quickly select content for any project by either using the default content or any other user defined content.

To define content as the default template, select report content from the List of all Sections and click on **Save as Default**.

To define content as a user defined template, select report content from the List of all Sections and click on Save Selection. You are asked to enter a name for your selection. This name appears then in the drop down box in the **User Selections** frame.

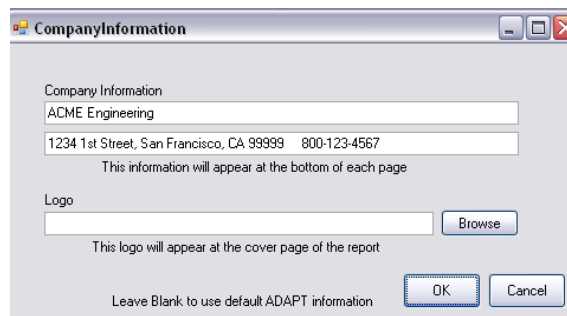



FIGURE 3.3-2

To open the “PT Summary Report” (**Fig. 3.3-3**) either click the **PTSum** button  on the tool bar or select the **PT summary** item on the *View* menu.

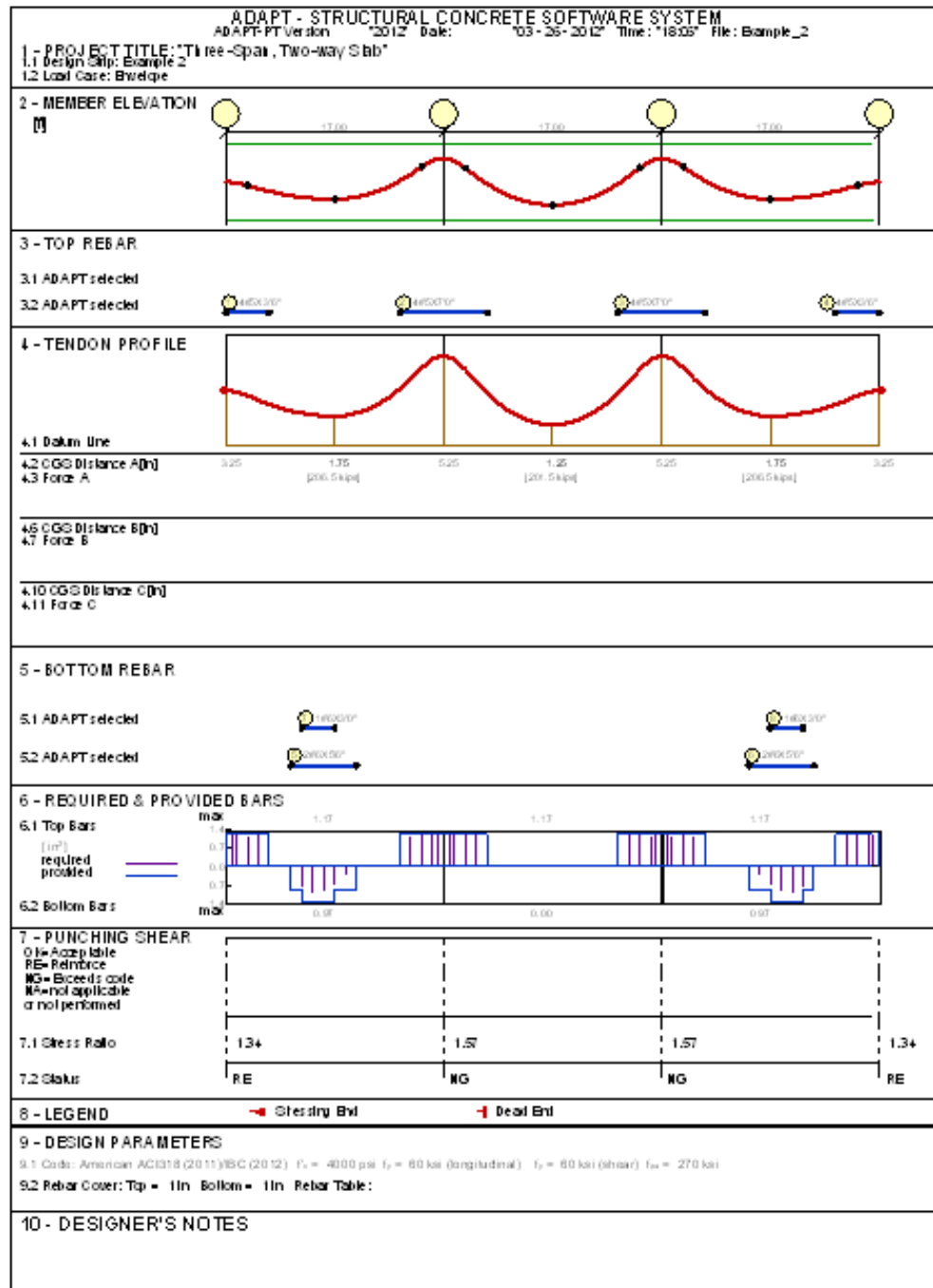


FIGURE 3.3-3

To view the graphs, either click the **Show Graphs** button  from the toolbar or select **Graphs** in the menu.

4 BEAM FRAME

Note that all new features described in the One-way Slab Example, unless noted otherwise, apply to beam frame system models as well. Refer to Chapter 2 for new feature descriptions as they are not repeated in this chapter.

The objective of this tutorial is to demonstrate the step-by-step procedure of ADAPT-PT 2012 to model, analyze and design a three-span flanged beam frame using grouted tendons. The structure represents a typical parking structure beam with its associated beam tributary and one-way slab construction. The procedure outlined in the tutorial is equally applicable to unbonded systems. The focus of the tutorial is to highlight the following aspects of the program:

- Use of bonded (grouted) post-tensioning;
- Automatic calculation of stress losses due to tendon friction and seating (draw in), creep, shrinkage, elastic shortening and relaxation in prestressing;
- Application of “effective width” in post-tensioned flanged beams;
- Adjustment of tendon force and profile to optimize the design;
- Design based on selection of number of strands, as opposed to “effective force”.

The geometry, material properties, loading and other features of the structure are given in the following. **Figure 4-1** shows the general layout of the structure.

(i) Material Properties

4.1 Concrete:

Compressive strength, f'_c	= 4000 psi	(27.58 MPa)
Weight	= 150 pcf	(2403 kg/m ³)
Modulus of Elasticity	= 3604 ksi	(24849 MPa)
Age of Concrete at stressing	= 5 days	
Compressive strength at stressing, f'_{ci}	= 3000 psi	(20.68 MPa)

○ Prestressing:

Low Relaxation, bonded System

Strand Diameter	= ½ in	(13 mm)
Strand Area	= 0.153 in ²	(98 mm ²)
Modulus of Elasticity	= 28000 ksi	(193054 MPa)
Coefficient of angular friction, μ	= 0.2	
Coefficient of wobble friction, K	= 0.0002 rad/ft	(0.0007 rad/m)

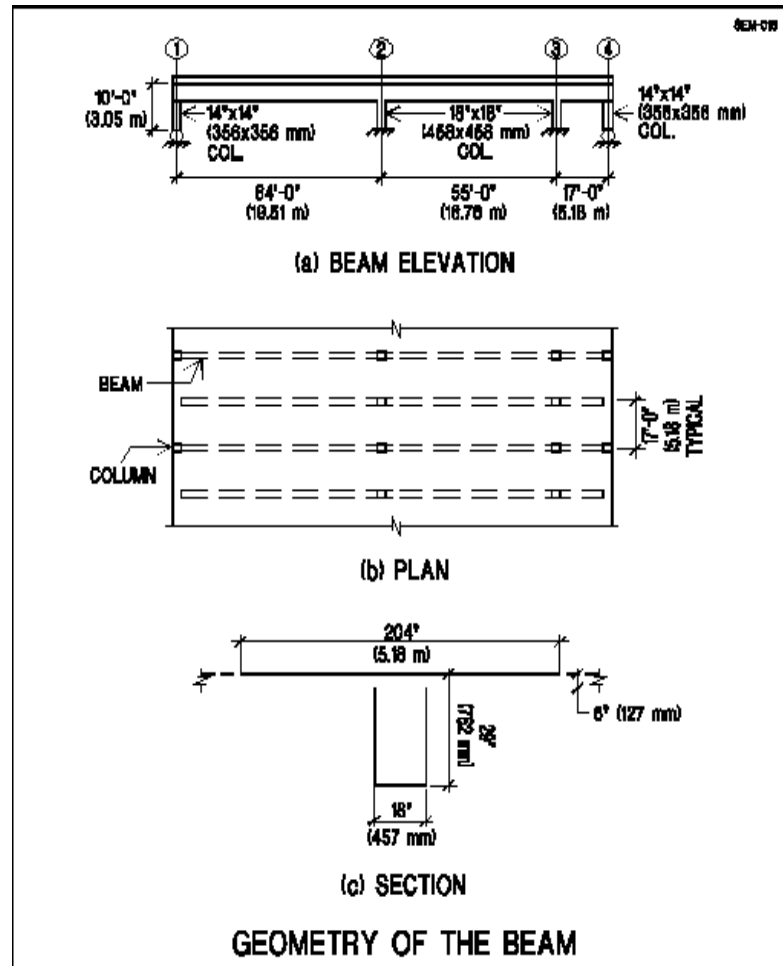


FIGURE 4-1

Ultimate strength of strand, f_{pu}	= 270 ksi	(1862MPa)
Ratio of jacking stress to strand's ultimate strength	= 0.8	
Anchor set	= 0.25 in	(6.35 mm)
Volume to surface ratio (V/S)	= 3.31 in	(84 mm)
Minimum strand cover		
From top fiber	= 2 in all spans	(50.8 mm)
From bottom fiber	= 3 in all spans	(76.2 mm)
○ Non-prestressed Reinforcing:		
Yield stress f_y	= 60 ksi	(413.69 MPa)
Modulus of Elasticity	= 29000 ksi	(199,949 MPa)
Minimum Rebar Cover	= 2 in Top	(50.8 mm)
	= 3 in Bottom	(76.2 mm)
(ii) Loading		
Dead load	= self weight + 0.29 k/ft (superimposed)	
Live load	= 0.54 k/ft	(7.88 kN/m)

4.2 GENERATE THE STRUCTURAL MODEL

In the ADAPT-PT 2012 screen, click the *Options* menu and set the default code as **ACI318-2011** and the *default units* as **American**.

4.2.1 Edit the project information

4.2.1.1 General Settings (Fig. 4.2-1)

Create the new project by clicking either **New** on the *file* menu or the **New Project** button on the toolbar. This automatically opens the *General Settings* input screen, as shown in **Figure 4.1-1**. You can enter the “General Title” and “Specific Title” of the project. For the purpose of this tutorial, enter the *General title* as **Three-Span T-Beam**. This will appear at the top of the first page of the output . Enter the *Specific title* as **Example 3**. This will appear at the top of the each subsequent page of the output.

Next, select the *Structural system* as **Beam**. Then you will be given an option of considering the *Effective Width in Bending*. In this case select **Yes**. For precompression (axial force), the entire tributary width is considered except for bending effects, where a limited width according to the ACI code is used.

Next, select the *Geometry Input* as **Conventional**. Segmental input is used for entering non-prismatic structures, i.e., those where the tributary width or the depth of the section changes within a span.

Click **Next** on the bottom right of this screen to open the input screen, *Design Settings*.

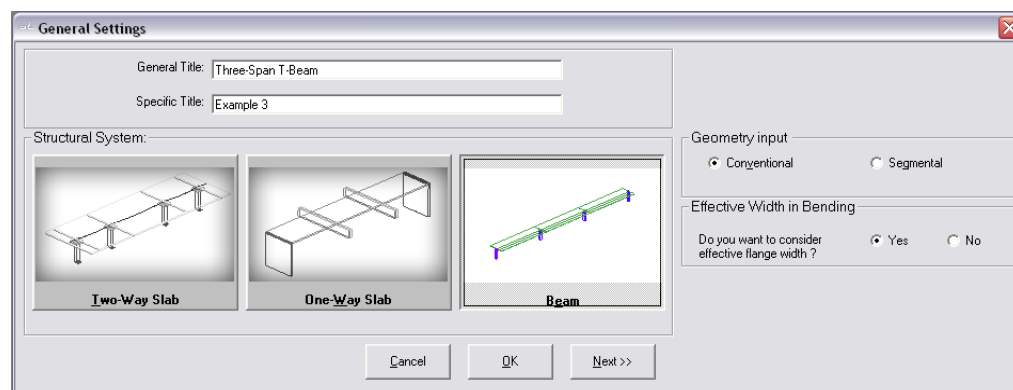


FIGURE 4.2-1

4.2.1.2 Design Code (Fig. 4.2-2)

In the second step you can specify the design code. PT 2012 now includes **ACI318-2011/IBC 2012**, which was selected at the onset of the new model. This should be selected in the Design Code Screen as shown in **Fig. 4.2-2**.

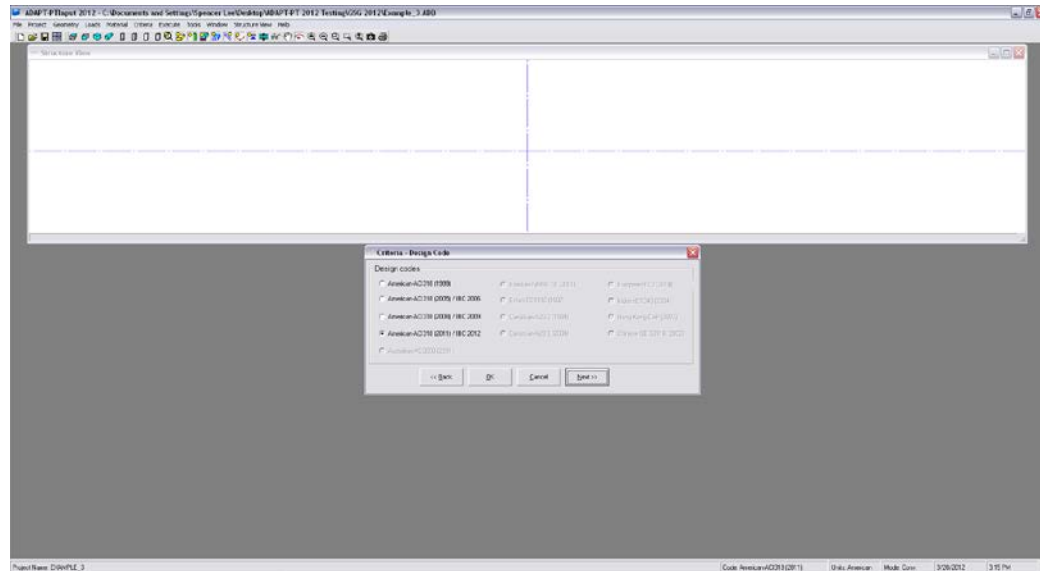


FIGURE 4.2-2

4.2.1.3 Design Settings (Fig. 4.2-3)

This screen is divided into two parts: *Analysis options* and *Design options*.

In Analysis options, you can select various calculation and design settings. First, select the *Execution Mode* as **Interactive**. In this mode, you have the opportunity to optimize the design by adjusting the tendon forces and the tendon drapes for each span in the “Recycle window”. This will be explained later in this section.

Next, select **Yes** for the *Reduce Moments to Face-of-Support* option. This option indicates that the calculated centerline moments at each support are adjusted to the face-of support. In addition to the centerline moments, PT 2010 prints out the moments reduced to the face-of-support.

Select **No** for the option to *Redistribute moments*.

For a beam system, the Equivalent Frame method is not applicable. You have the option to *Increase the Moment of Inertia* over the supports. This option will cause the program to use a larger moment of inertia over the supports than given by the cross-sectional geometry of the beam. This, in turn, affects the relative

distribution of the moments and may affect the amount of post-tensioning required. For this tutorial, select **No**.

In *Design options*, you can either *Use all provisions of the code* that you selected in the previous step, or *Disregard the following provisions* such as *Minimum rebar for serviceability*, *Design capacity exceeding cracking moment*, and *Contribution of prestressing in strength check*. For this tutorial, keep the default setting and use all code provisions.

The choice to *Include (DL + 0.25LL) case of UBC* is now part of the *Design options* in this step. Selecting this option allows you to enter the *Ratio of reduced live load to actual live load*. Leave this option unchecked for this example. This is a UBC (Uniform Building Code) requirement (not required by ACI318-2011 and IBC 2012) used to determine the amount of mild steel reinforcement for one-way slab systems and beams, when reinforced with unbonded tendons. This is not applicable for this tutorial since grouted, bonded tendons will be used.

ADAPT-PT 2012 includes the option to *Generate moment capacity based on Design Values* or *User-Entered Values*. If **Design Values** is selected, the program will calculate and report positive and negative moment capacities based on prestressing steel, base reinforcement as defined by the user (this is discussed later in this section) and program-calculated reinforcement. Demand moments at 20th points along each span are also reported. When **User-Entered Values** is selected, the program will calculate and report similar moment capacities and demand moments where the capacities are based on prestressing steel and based reinforcement as defined by the user. For this example, select **Design Values**.

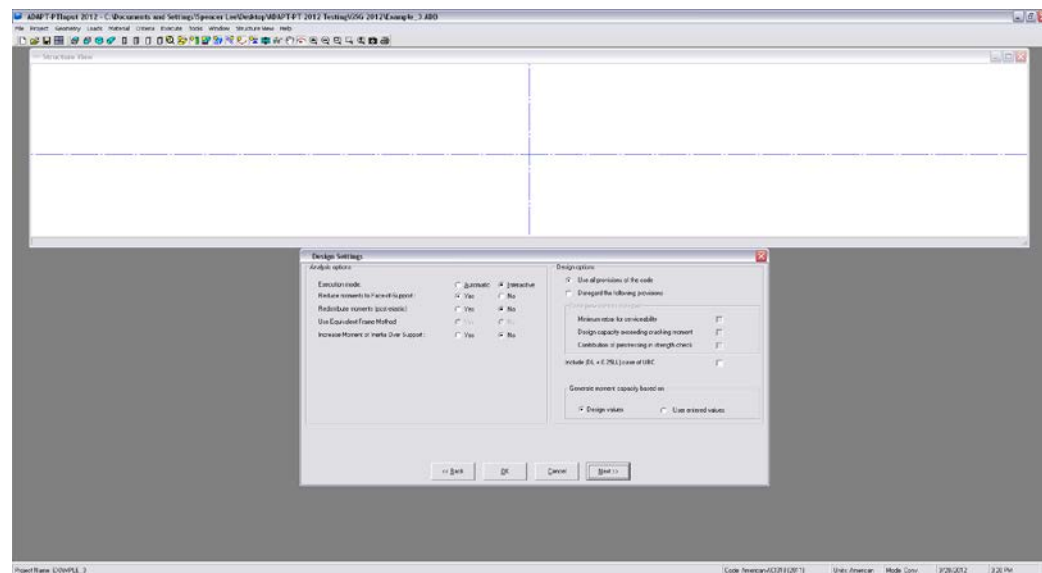


FIGURE 4.2-3

Click **Next** at the bottom right of the Design Settings screen to open the *Span Geometry* input screen.

4.2.2 Edit the geometry of the structure

4.2.2.1 Enter Span Geometry (Fig. 4.2-4)

This screen is used to enter the cross-sectional geometry of the slab at mid-span.

Set the *Number of Spans* as **3** either by clicking **up arrow** or using **CTRL +**.

Next, enter the dimensions. All dimensions are defined in the legend at the top of the screen and/or illustrated in the appropriate section figure. The section type for any span can be changed by clicking on the button in the Section (Sec) column.

For the first span select the section, *Sec*, as **T-Section**, edit **64** ft (19.51 m) for length (*L*), **18** inches (457 mm) for width (*b*), **34** inches (864 mm) for height (*h*), **204** inches (5182 mm) (tributary width) for width of the flange (*b_f*) and **5** inches (127 mm) (thickness of the slab) for *h_f*. Repeat the same procedure for span 2 and span 3 by changing the values as shown in **Figure 4.2-4**.

You can use the “Typical” input row (top row) to enter similar dimensions. To enter typical values, type the **value** into the appropriate cell in the top row and then press **enter**. The typical value will be copied to all the spans.

As you enter the values, the span is displayed in real-time in the 3D window. You can zoom in and out in the *Structure View* with the help of your mouse wheel or with the help of the *Zoom In* or *Zoom Out* buttons in the *View Toolbar*.

You can access special data editing options by selecting data cells and right clicking. Available options include Insert New Line, Delete Line.

The Reference height (*Rh*) identifies the position from which the tendon height is measured. Typically, the reference line is selected to be the soffit of the member. Hence, for this tutorial, select beam depth. Click **?** with the **Rh** definition in the Legend box to learn more about this. Type the Reference height, *Rh* as **34** inches (864 mm), i.e., depth of the beam, for all spans.

The Left and Right Multiplier columns (<-M and M->) are used to specify the tributary width to indicate how much of the tributary falls on either side of the frame line. Tributary widths can be specified using either the “Unit Strip” method or the “Tributary method”. Enter **0.50** for both the left and right multipliers since equal tributary falls on either side of the frame line.

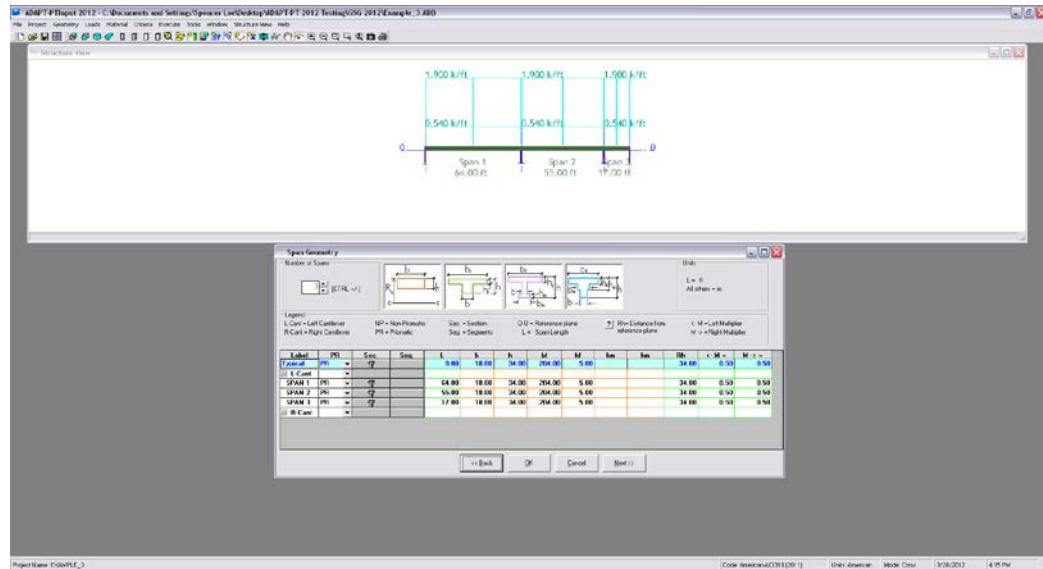


FIGURE 4.2-4

Click **Next** on the bottom to open the next input screen, *Effective Flange Width*.

4.2.2.2 Enter Effective Flange Width (Fig. 4.2-5)

In the *General Settings* input screen, we selected “yes” to include effective flange width; therefore the screen as shown in **Figure 4.2-5** opens. In this screen, the default values of “**b_e**” are calculated from the geometry according to the ACI code. You cannot modify these values. If you want to input these values, change *Effective width calculation method* option to **User input**.

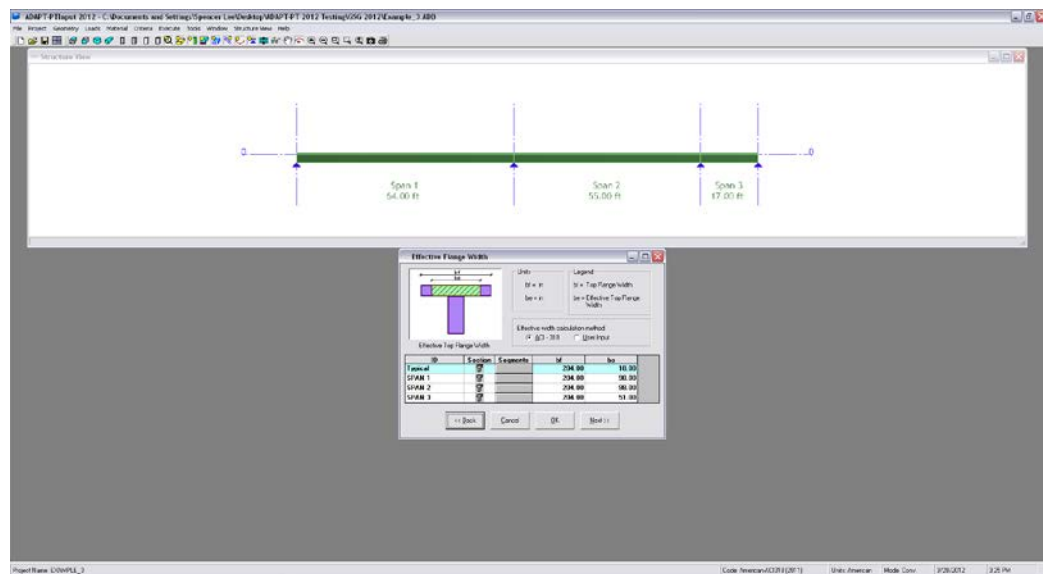


FIGURE 4.2-5

Click **Next** on the bottom to open the next input screen, *Support Geometry*.

4.2.2.3 Enter Support Geometry (Fig. 4.2-6)

This screen is used to input column or wall heights, widths and depths. You may enter dimensions for columns/walls above and/or below the slab.

Select **Lower column** from the *Support selection* box and enter **10 ft (3.05m)** for *H1* in the “Typical” row (top row). Press **ENTER** to assign this value to all the lower columns.

Next, enter the dimensions of the supports. B is the dimension of the column cross- section normal to the direction of the frame. D is the column dimension parallel to the frame. Enter the column dimensions as in **Figure 4.2-6**.

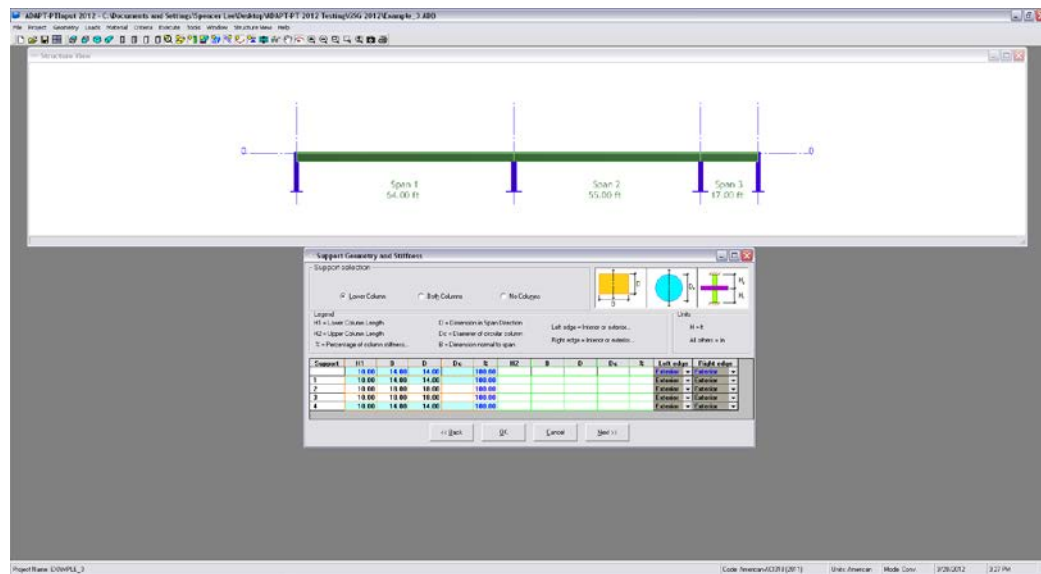


FIGURE 4.2-6

Click **Next** on the bottom line to open the input screen, *Supports Boundary conditions*.

4.2.2.4 Enter Supports Boundary Conditions (Fig. 4.2-7)

This screen is used to enter the support widths and column boundary conditions.

Support widths are only entered if you answered **Yes** to the *Reduce Moments to face-of-support* question on the *Design Settings* screen. If you answered **No**, you cannot input the value in the SW column. This input value will be used to calculate the reduced moments.

Since the support width SW is set to the column dimension (D) as a default, the SW values will be automatically determined from the support geometry and cannot

be modified by the user. If you want to input the *SW* values, **uncheck** the *SW=Column Dimension* box.

Select *LC (N)*, boundary condition for the near end, as **1(fixed)** from the drop down list. *LC (F)*, boundary condition for far end, as **2(hinged)** for the first and last supports, and **1(fixed)** for the second and third supports.

Leave the *End Support Fixity* for both the left and right supports as the default **No**. This will be used when the slab or beam is attached to a stiff member.

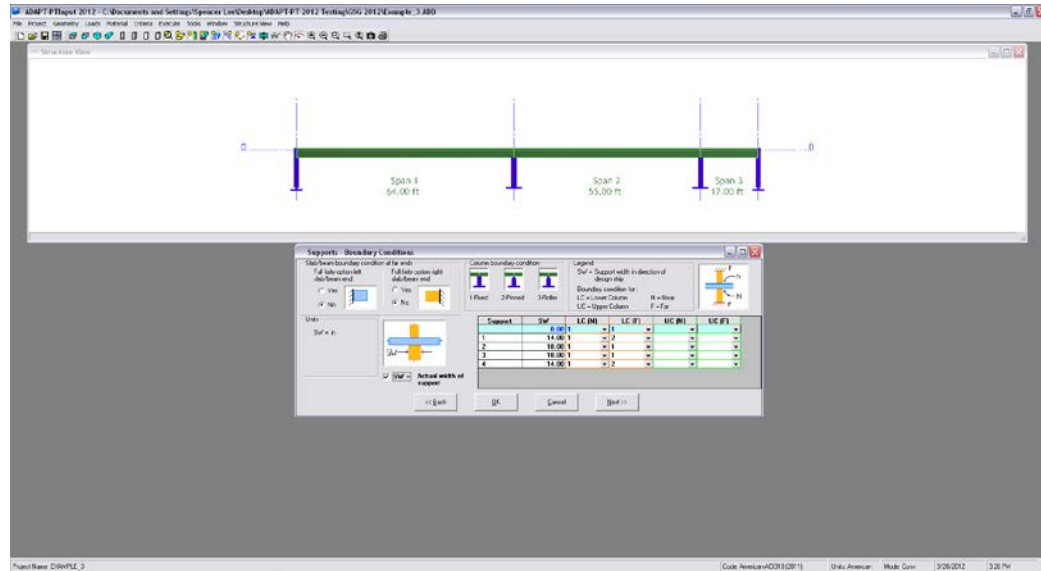


FIGURE 4.2-7

Click **Next** at the bottom of the screen to open the input screen, *Loading*.

4.2.3 Enter Data

4.2.3.1 Edit the loading information (Fig. 4.2-8)

Any number of different loads and load types may be entered for a span.

Load types available in PT 2012 are: *Uniform, Partial Uniform, Concentrated, Moment (Concentrated), Line, Triangular, Variable and Trapezoidal*.

Enter the span number as **1** in the *Span* column. If the loads are the same for all the spans, you can type **ALL** or **all** in the *Span* column. This will copy the data to all the spans.

If you choose not to include Self-weight, you now have the option to define the self-weight (**SW**) as a *Class*. In any case, you can choose to specify additional Dead Load as Superimposed Dead Load (**SDL**) as a *Class*. PT 2012 gives you the option to specify any other load (i.e. hydrostatic, soil, etc.) in the **X** *Class* loading.

Select the *Class* as **SDL** from the drop down list and specify the load type as line either by typing **L** in *L-?* or by **dragging the icon** from the graphics of the line loads.

Edit **1.90 k/ft (27.73 kN/m)** for dead load in the *PI* column. This loading includes both superimposed dead load and self-weight. Since the program can calculate self-weight automatically, you must answer **No** to the *Include Self-Weight* question at the top right of the screen. Note that the entry for **unit weight** will become grayed out. If you enter **L-L**, you have to enter *a* (starting point of loading from the left support), and *b* (end point of loading from the left support).

Repeat the procedure in the second row by changing *Class* to **LL** and the *PI* value to **0.54 k/ft (7.88 kN/m)**.

Answer **No** to *Skip Live Load?* at the top right of the screen.

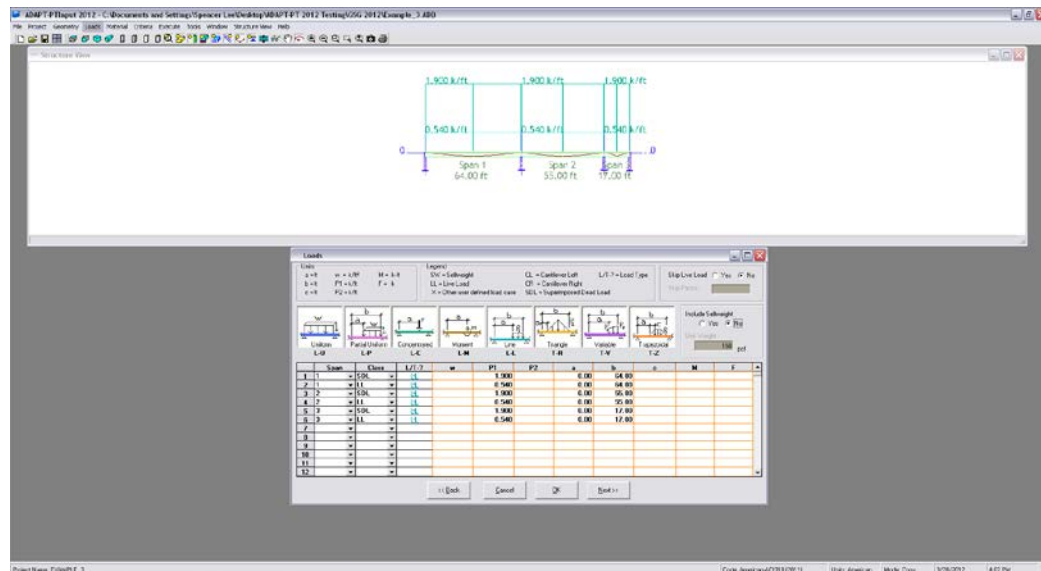


FIGURE 4.2-8

Click **Next** at the bottom of the screen to open the *Material-Concrete* input screen.

If you entered *Span* as **all** and selected uniform load, click **Back** and go back to the loading screen. You can see that all the loads are copied to the individual spans as in **Figure 4.2-8**.

4.2.4 Edit the material properties

4.2.4.1 Enter The Properties Of Concrete (Fig. 4.2-9)

Select **Normal Weight** and **enter** the *Strength at 28 days* for slab/beam and column. When you press **enter** from the strength input value, the *Modulus of Elasticity* will be calculated automatically based on the concrete strength and the appropriate code formula.

For this tutorial, keep the default values for concrete strength and the creep coefficient. The creep coefficient will be used in the calculation of long-term deflection.

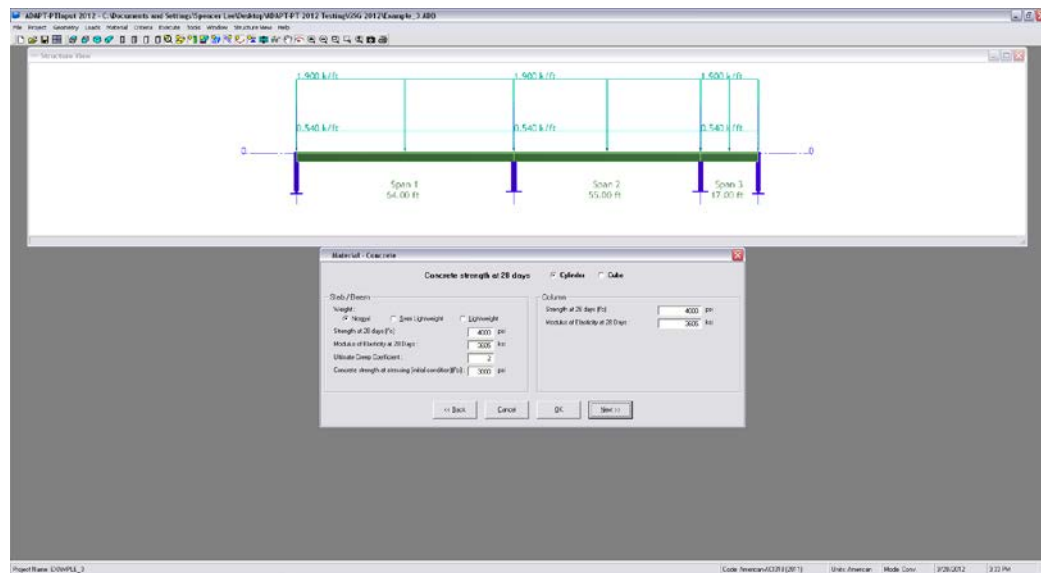


FIGURE 4.2-9

Click **Next** at the bottom of the screen to open the next input screen, *Material Reinforcement*.

4.2.4.2 Enter The Properties Of Reinforcement (Fig. 4.2-10)

The screen is divided into two parts: *Longitudinal reinforcement* and *Shear reinforcement*.

For this tutorial, keep the default values for *Yield Strength* and *Modulus of Elasticity*. Change the *Preferred Bar Size* for *Top* and *Bottom* to **8** and **8** (25, 25). These will be used when calculating the number of bars required.

Change the Preferred Stirrup Bar Size to **4** (13).

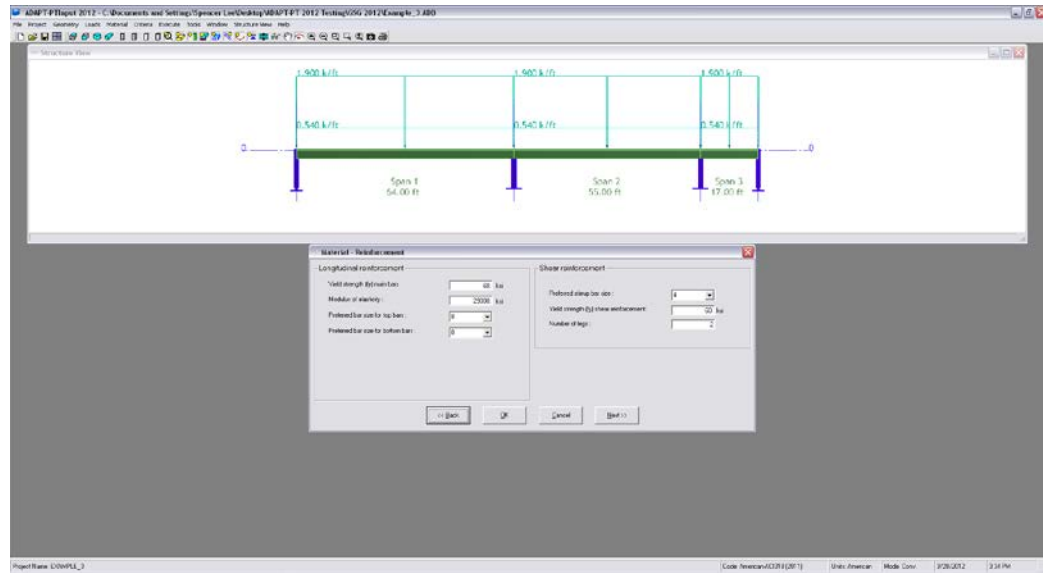


FIGURE 4.2-10

Click **Next** at the bottom of the screen to open up the next screen.

4.2.4.3 Enter The Post-Tensioning System Parameters (Fig. 4.2-11)

Select the *Post-tensioning system* as **Bonded** and leave the default values for the other properties.

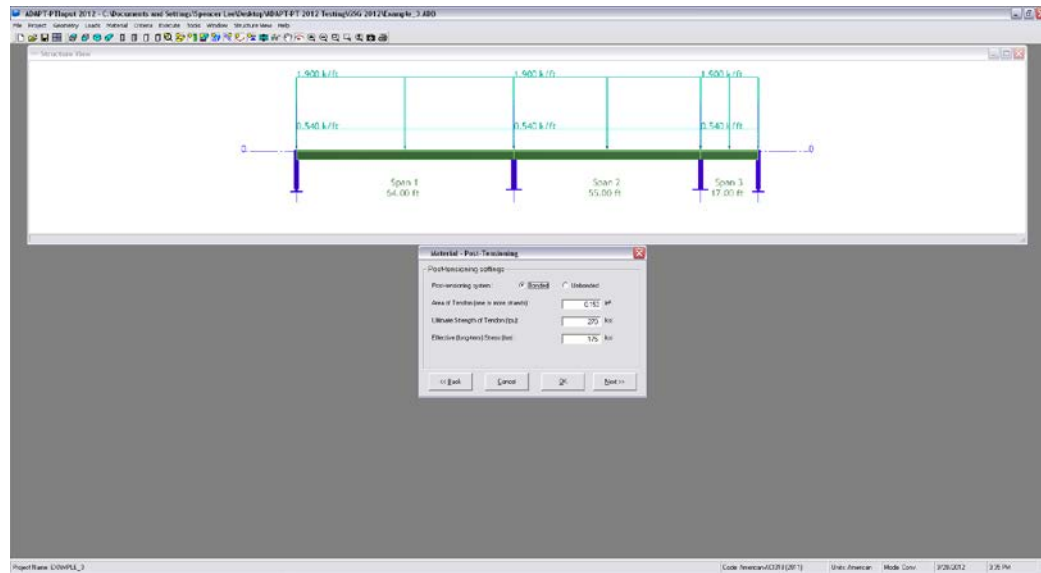


FIGURE 4.2-11

The “effective (long-term) stress” entered will be used as the initial value in the iterative procedure of the program. The program will determine the applicable stress at any point based on the long-term loss parameters you entered.

Click **Next** at the bottom of the screen to open the next input screen.

4.2.4.4 Edit Base Reinforcement (Fig. 4.2-12)

The program allows you to specify a base reinforcement that is taken into consideration when designing the structure. The base reinforcement can be input as mesh or isolated rebar. For this example, base reinforcement will not be considered. Select **No** in the *Base Reinforcement* section.

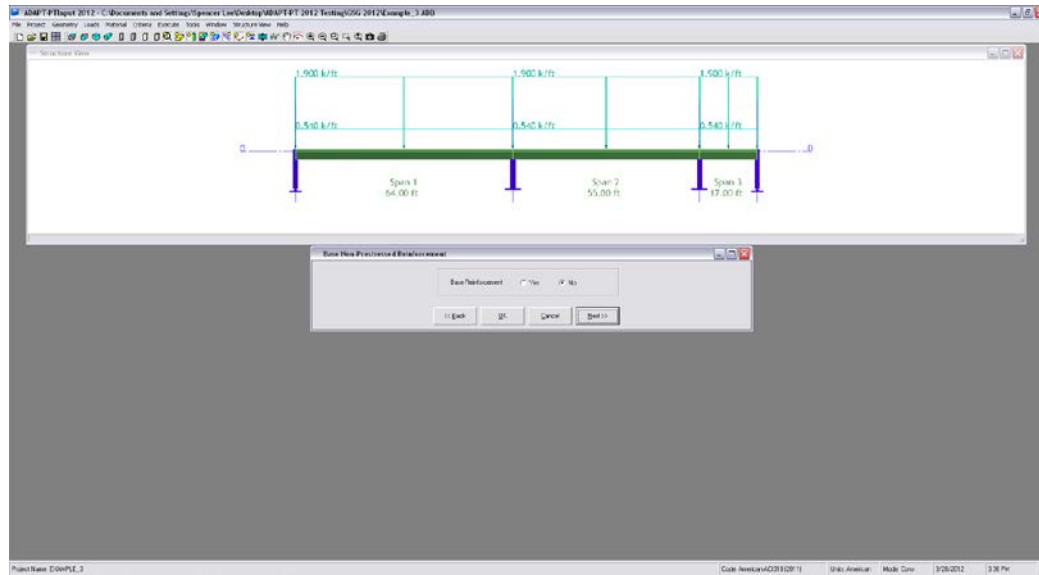


FIGURE 4.2-12

4.2.5 Edit the design criteria

4.2.5.1 Enter The Initial And Final Allowable Stresses. (Fig. 4.2-13)

Tensile stresses are input as a multiple of the square root of f'_c , and compressive stresses are input as multiple of f'_c .

Change the top and bottom final tensile stress to $9\sqrt{f'_c}$ ($0.75\sqrt{f'_c}$). This is the allowable tensile stress per ACI 318-2011 for a Class 'T' system (transition zone between an un-cracked and cracked section).

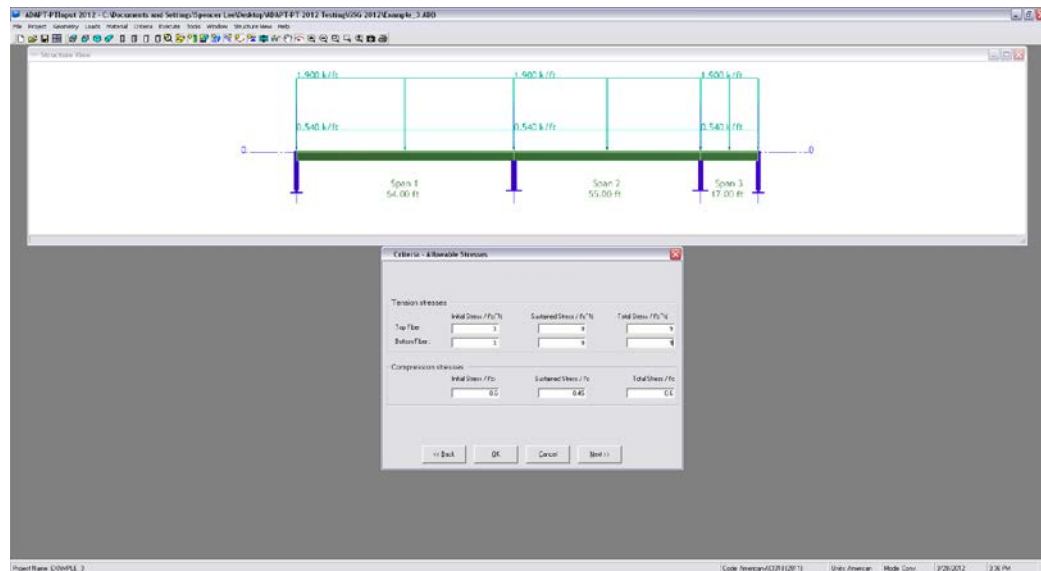


FIGURE 4.2-13

Click **Next** at the bottom of the screen to open the input screen, *Criteria – Recommended Post-Tensioning Values*.

4.2.5.2 Enter The Recommended Post-Tensioning Values (Fig. 4.2-14)

This screen is used to specify minimum and maximum values for average precompression (P/A : total prestressing divided by gross cross-sectional area) and percentage of dead load to balance (W_{bal}). These values are used by the program to determine the post-tensioning requirements and the status of the P_{min}/P_{max} and $WBAL Min/ Max$ indicators on the “Recycle” window.

The values given as default are according to the code and the experience of economical design. Keep the **default values**.

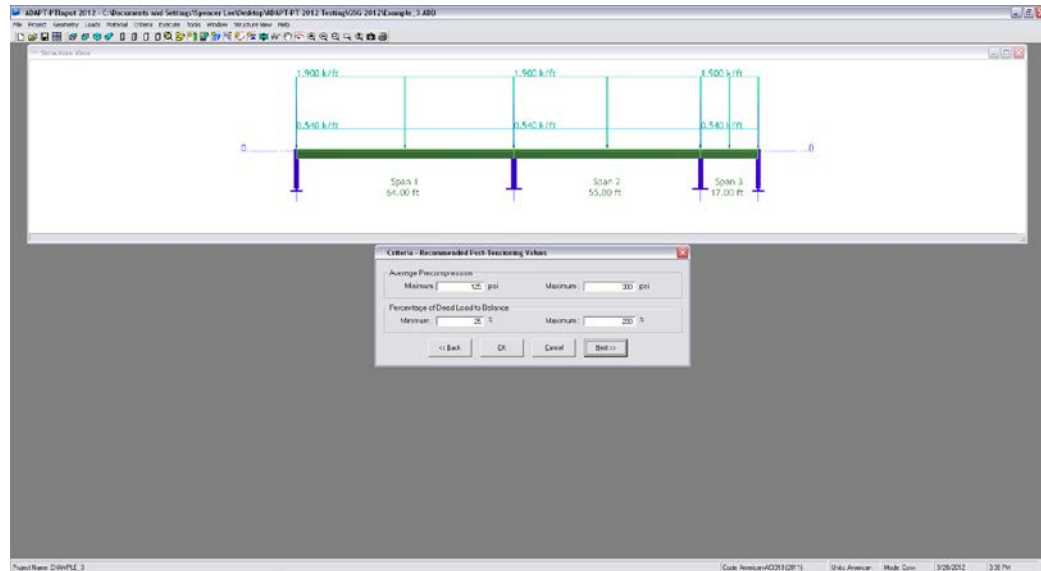


FIGURE 4.2-14

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Calculation Options*.

4.2.5.3 Select The Post-Tensioning Design Option (Fig. 4.2-15, 16)

The two design options are “Force Selection” and “Calculate force/number of tendons” as in **Figure 4.2-15**. “Force Selection” is the default option.

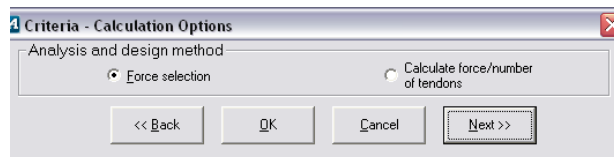


FIGURE 4.2-15

Select the **Calculate force/number of tendons** option. The screen will prompt for the information required to calculate the prestressing losses, as in **Figure 4.2-16**

To calculate friction stress losses, enter the information given in material properties, as in **Figure 4.2-16**.

Long-term losses may either be entered as a lump sum value or the program can calculate them using the provided information.

Select **Yes** to perform long-term loss calculation. Enter *Age of stressing* as **5** days and press **enter**. The *Strength* and *Modulus of Elasticity* at stressing will be calculated automatically by the program. However, if concrete strength at stressing is established through cylinder/cube tests, enter the test result. For most anchorage devices, there is a specified minimum concrete strength for stressing. In

this tutorial, the minimum value is 3000 psi (20.68 MPa) which is already entered in the *Material – Concrete* input screen.

Edit **80%** for *Relative Ambient Humidity (RH)* and **3.31** inches(84 mm) for *Volume to Surface Ratio (V/S)*. V/S is the calculated value from the given section dimensions.

Answer **No** to *Are all tendons stressed at one time* question. This information is used to determine the stress losses in prestressing due to elastic shortening of the member.

FIGURE 4.2-16

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Tendon Profile*.

4.2.5.4 Specify The Tendon Profiles (Fig. 4.2-17)

The program allows you to specify up to three tendon paths per span. You can define one profile for each of the three tendons.

In the section *Option for tendons* you can define the *Default extension of terminated tendon as fraction of span*. Also, you can specify the *Shape of tendon extension* from the *Left end* and the *Right end*. For this example, use the default values.

In this example we only use tendon A. From the *Type* drop down list, select **2** (Partial parabola) for spans 1 and 2, and **3** (Harped Parabola) for the third span. For the first span, change the inflection points ($X1/L$ & $X3/L$) to **0.031**. For second

span change $X1/L$ to **0.036** and $X3/L$ to **0**. Keep the low point ($X2/L$) at mid-span, i.e., at **0.5**.

New!

New to ADAPT-PT 2012 is the option to “Set tendon ends at CG of effective section.” If selected, this option will automatically locate the tendon end CG values at the centroid of the section based on effective properties rather than properties associated with the full tributary of the flange. Note that by selecting this option, the program will calculate equivalent moments at each end equal to the PT force * eccentricity from the CG of the full-section to the CG of equivalent section. For this example, leave this option unchecked.

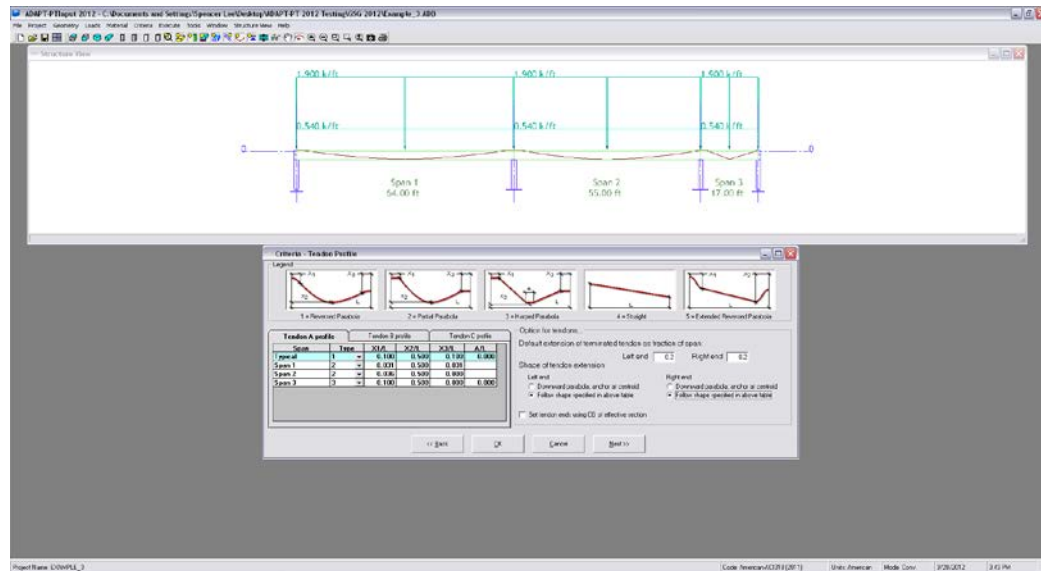


FIGURE 4.2-17

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Minimum Covers*.

4.2.5.5 Specify Minimum Covers For Post-Tensioning Tendons And Mild Steel Reinforcement (Fig. 4.2-18)

The cover for the prestressing steel is specified to the center of gravity of the strand (cgs). Therefore, for ½ inch (13 mm) strand, cgs is minimum cover + ½ * ½, i.e., cgs = cover + 0.25” (cgs = cover + ½ * 13). Edit *CGS* of tendon as **2.25** inches (57 mm) for the top fiber and **3.25** inches (44 mm) for the bottom fiber.

For *Non-prestressed Reinforcement*, edit **2** in (51 mm) *Cover* for the top and **3** in (76 mm) *Cover* for the bottom.

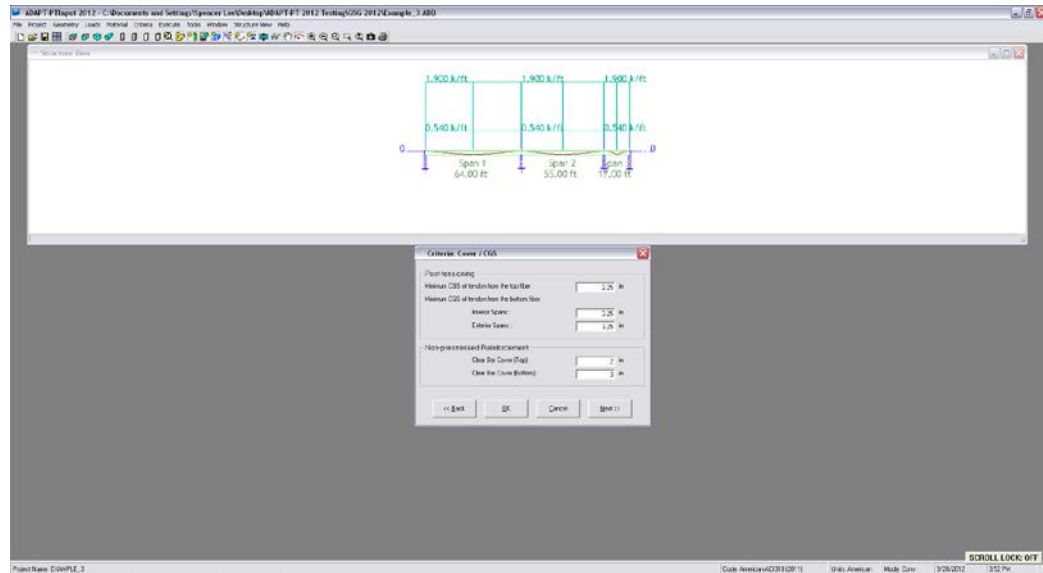


FIGURE 4.2-18

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Minimum Bar Extension*.

4.2.5.6 Specify Minimum Bar Length And Bar Extension Of Mild Steel Reinforcement (Fig. 4.2-19)

The values given are for development of bars required to supplement prestressing in the strength check design. These values may be modified if necessary. For this tutorial, keep the default values.

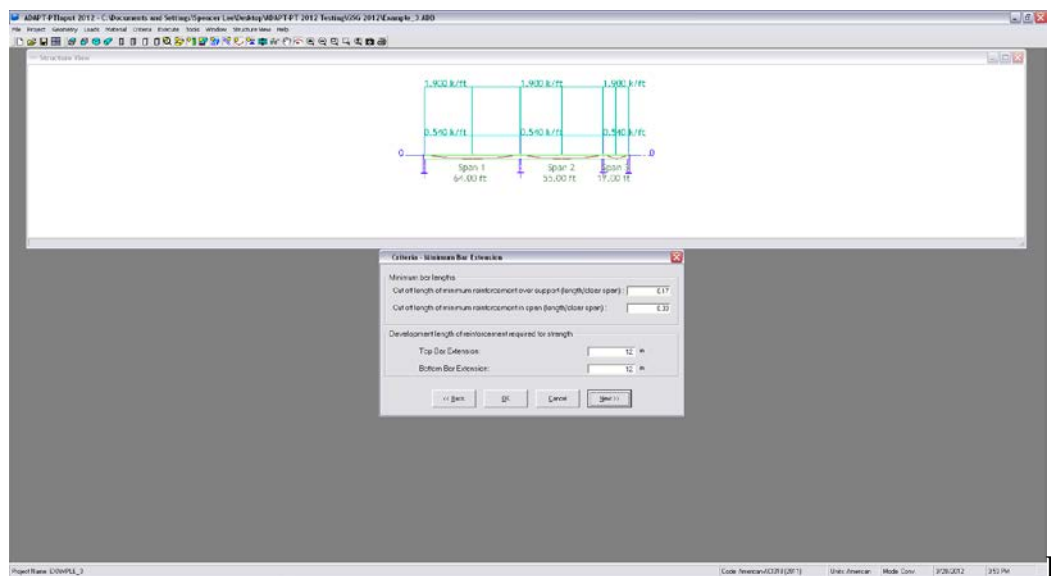


FIGURE 4.2-19

Click **Next** at the bottom of the screen to open the next input screen, *Load Combinations*.

4.2.5.7 Input Load Combinations (Fig. 4.2-20)

This screen is used to input the load combination factors for service and strength (ultimate) load conditions. It is also used to enter any applicable strength reduction factors. The default values are according to the ACI 318-2011. For this example, use the default values.

The program allows you to specify four strength load combinations and four service load combinations. For ACI 318-2011, two of the service load combinations are reserved for sustained load and two for total load. In the entry for the Initial load combination factors, set the factor for **SW** to zero and 0.85 for **SDL**. The reason for this is that the **SW** is included in the **SDL** and is 85% of the total DL applied.

ADAPT-PT 2011 allows lateral moments to be included and designed for in combination with gravity loads. To do this, select the check mark to *Include lateral loads* and click on the *Set Values* button to define *Lateral moments* (Fig. 4.1-21) and *Lateral load combinations* (Fig. 4.2-22). For this example, lateral loads will not be included.

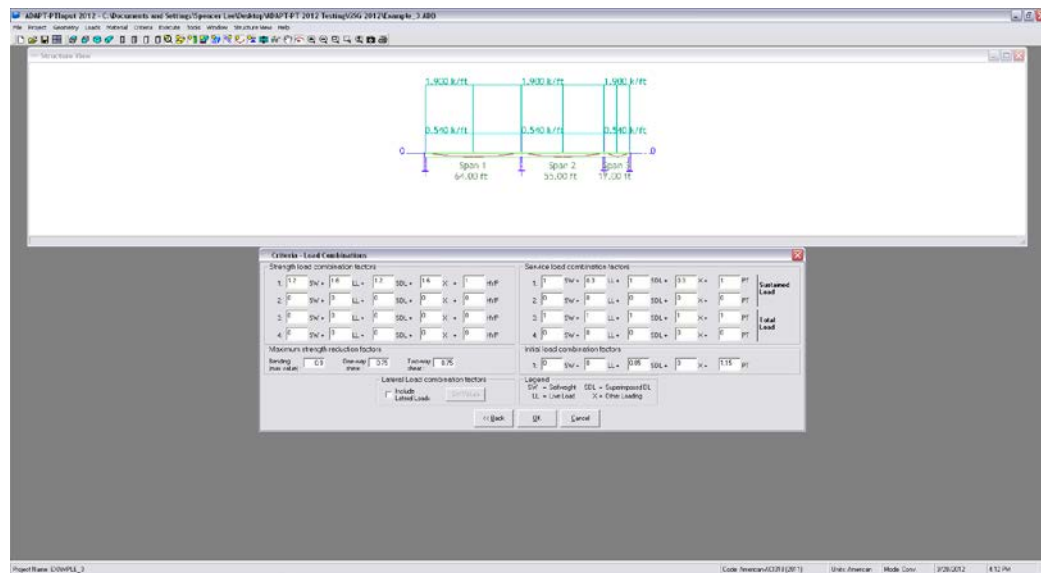


FIGURE 4.2-20

4 Lateral Input Data

Lateral load combination Lateral moments

Lateral Moments

Moments Units [k-ft]

Span	Left of Span	Right of Span
Typical		
1	0.000	0.000
2	0.000	0.000
3	0.000	0.000

positive direction shown

Legend
M1 = Left of span
M2 = Right of span

OK Cancel

FIGURE 4.2-21

Lateral Input Data

Lateral load combination Lateral moments

Load Combination Factors

1) U = 1.20 SW + 1.00 LL + 1.20 SDL + 1.00 X + 1.00 Sec + 1.00 Lat

2) U = 0.90 SW + 0.00 LL + 0.90 SDL + 0.00 X + 1.00 Sec + 1.00 Lat

Options

Do lateral moments change sign ☐ No ☒ Yes

PT to resist Factored Moments: 100


Legend
SW = Selfweight
LL = Live Load
SDL = Superimposed DL
X = Other loading
Lat = Lateral (Seismic/wind)
Sec = Secondary

OK Cancel

FIGURE 4.2-22

Click **OK** at the bottom of the screen to finish the input wizard.

4.3 SAVE AND EXECUTE THE INPUT DATA

To save the input data and execute the analysis, either select **Execute Analysis** from the *Action* menu of the menu bar or click on the **Save & Execute Analysis** button . Then, give a **file name** and **directory** in which to save the file. The program saves all sub-files in a specific folder with the name selected by the user, along with the .adb file at the user-defined directory. Once the file is saved, the program will automatically execute the analysis by reading the data files and performing a number of preliminary data checks.

Once the execution has successfully finished, i.e., without any errors, the “PT Recycling” window shown in **Fig 4.3-1** opens. If an error is detected, the program will stop and display a message box indicating the most likely source of the error.

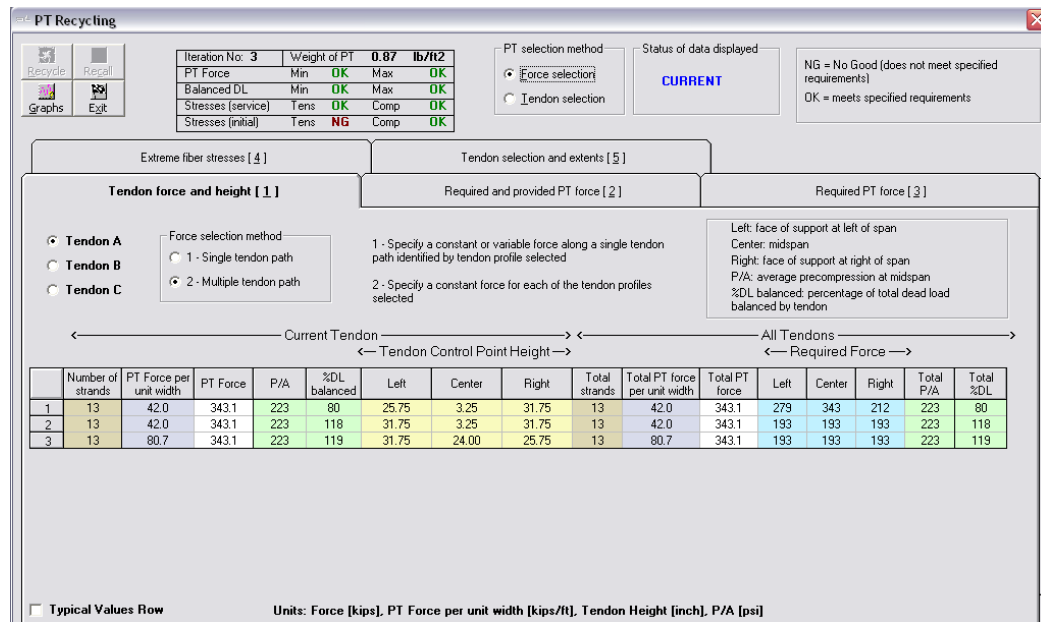


FIGURE 4.3-1

Here you can choose either “Force Selection Mode” or “Tendon Selection”, since we selected “Force/Tendon Selection” during input. Select **Tendon Selection**, and go to the **Tendon selection and extents [5]** tab as shown in **Figure 4.3-2**. In this mode, the actual number of strands, as opposed to effective forces, may be specified. Note that new enhancements to tendon selection options are detailed in Example 1.

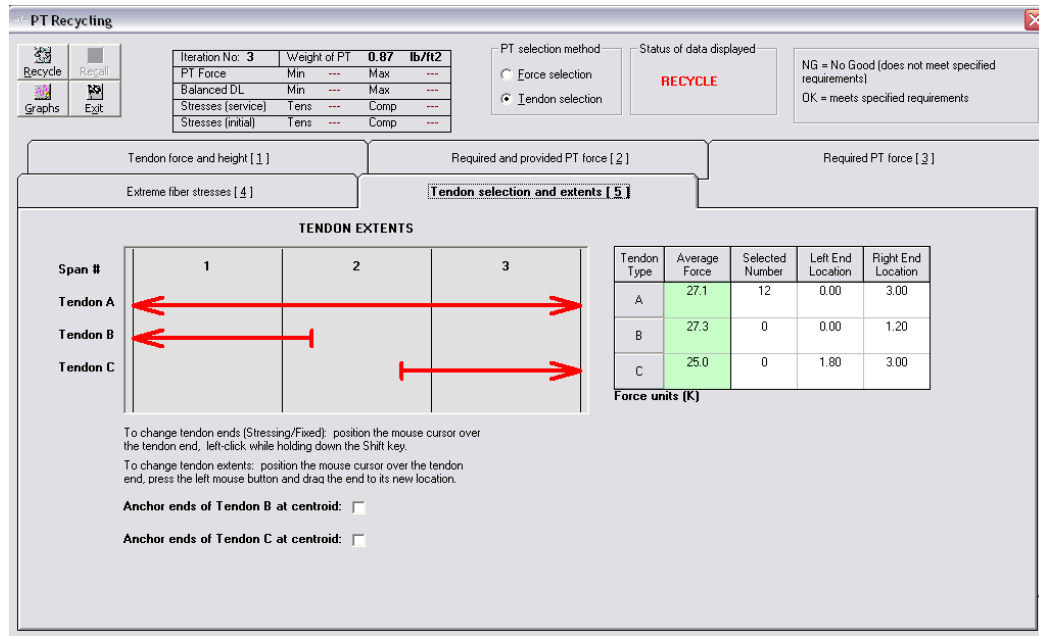


FIGURE 4.3-2

Here you can edit the post-tensioning layout by adjusting the tendon profiles, editing the number of strands in a tendon type, and changing the stressing ends and/or extent of the tendons to arrive at an acceptable solution.

To change the tendon heights, click the **Tendon force and height [1]** tab (Fig. 4.3-3). Note that when the “Tendon Selection” option is active, the user cannot access the *Force* column on this tab. In the “Tendon Selection” mode, forces are calculated based on the number of strands and the final stresses in the strand.

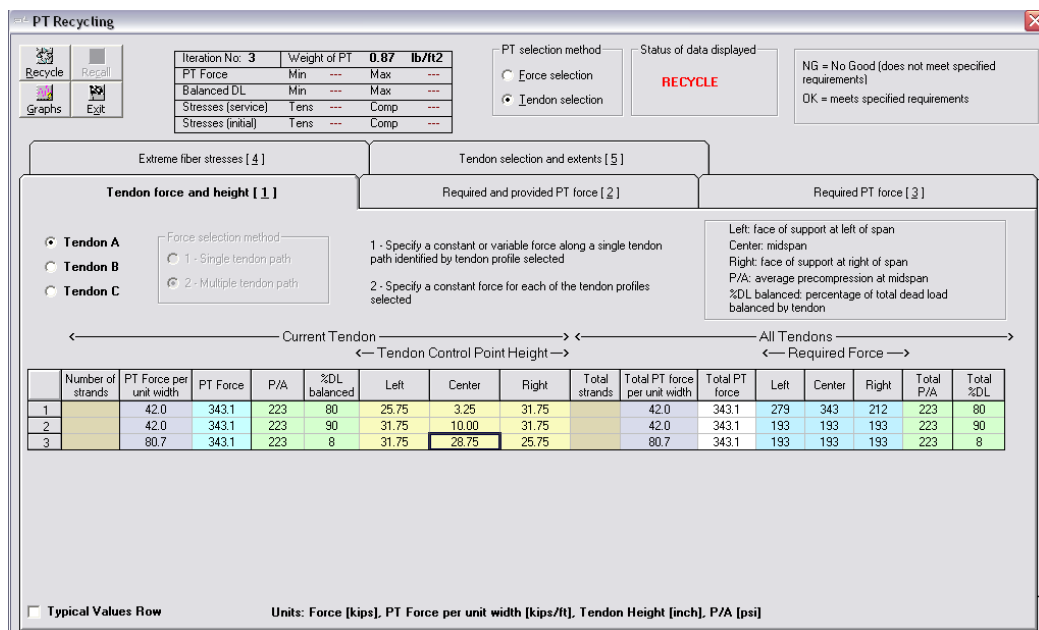


FIGURE 4.3-3

Change the second span *Center* height to **10** inches (254 mm) (since this span is shorter than the first, balancing a lower percentage of self-weight is beneficial) and third span *Center* height to **28.75** inches (730 mm) (average of the left and right heights, to make a straight profile).

To edit the number of strands in a tendon type, click the **Tendon selection and extents [5]** tab (Fig. 4.3-4). The number of strands to use for each tendon type is shown in the “Selected Number” column. These numbers may be changed independently of one another. To delete a tendon type, set the number of strands selected to zero. To add a tendon type, enter the number of strands to use for that type. The “Average Force” column shows the average force in each strand. Change the number of strands for *Tendon Type A* to **8** and for *Type B* to **4** (Fig. 4.3-4).

To change a tendon end from dead to stressing or stressing to dead, hold down the **Shift** key and **left click once** at the end of the tendon. Clicking a second time will change the tendon back to its original configuration. Note that the tendon must have at least one stressing end. To change the extent of *Tendon Type B* or *Tendon Type C*, position the **cursor** over the tendon end, hold down the **left mouse button** and **drag the end** to the desired location. Change the right end of the *Tendon Type A* from stressing to **dead**.

Next, Click **Recycle** at the top left of the screen to recalculate the force provided. There is no limit on the number of changes that can be made or the number of times the window can be recycled. After recycling, the window as in Fig. 4.3-5 opens.

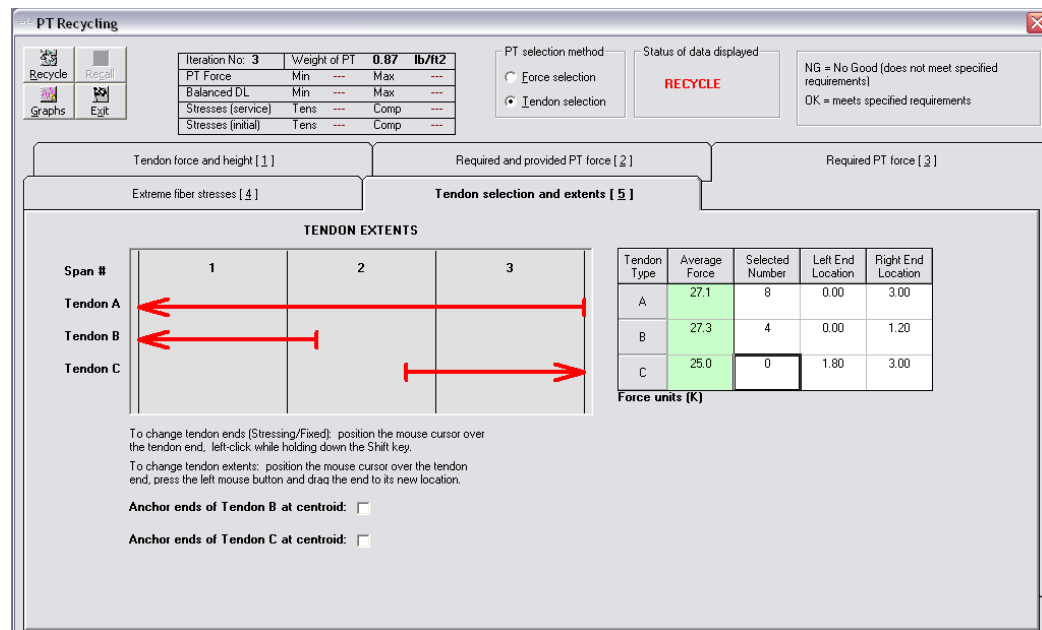


FIGURE 4.3-4

In the second cycle, the weight of post-tensioning was 870 psf, now it's 710 psf and all the indicators show "OK".

You can check the final stresses for the **Total load condition** either by clicking the **Extreme fiber stresses [4]** tab (Fig. 4.3-6) or by clicking **Graphs** at the top left of the screen.

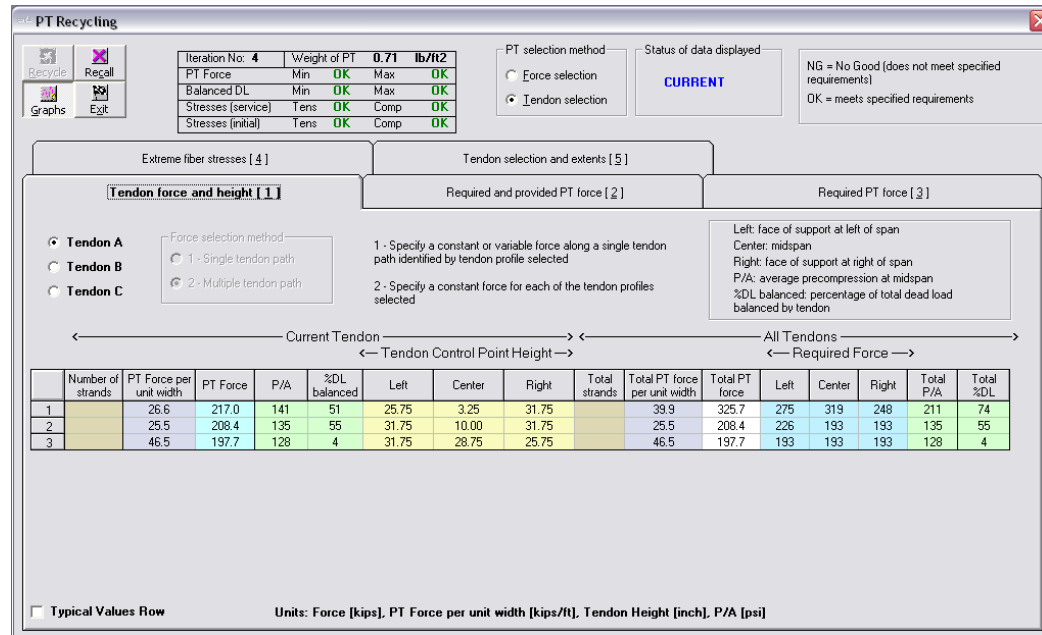


FIGURE 4.3-5

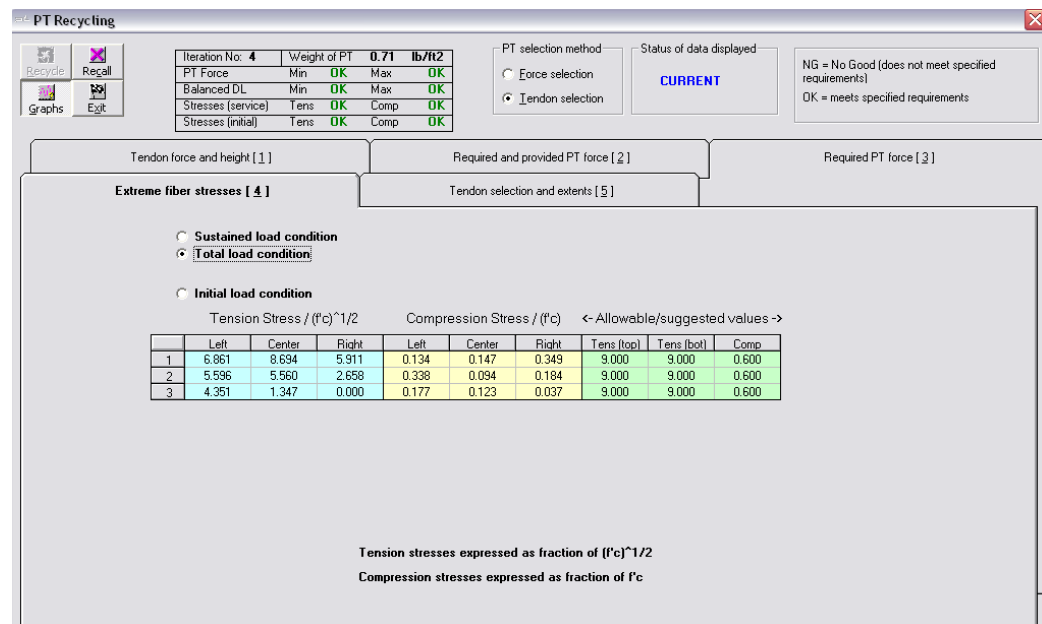


FIGURE 4.3-6

Graphs displays a set of three graphs which provide detailed information on the tendon profile, the tension and compression stresses and the required versus provided post-tensioning forces at 1/20th points along the spans (**Fig. 4.3-7**).

The top diagram, the **Tendon Height Diagram** shows the elevation of the tendon profile selected. The tendon profile can be viewed either with or without the concrete outline by checking the option at the left of the screen.

The second diagram, **Stress Diagrams**, plots the maximum compressive and tensile stresses at the top and bottom face of the member. You can view the stresses due to *Dead Load*, *Live Load*, *Post-tensioning* and *Service Combination* each separately, or in combination, by selecting the options at the screen. Also you can verify the top and bottom stresses due to service combination with the allowable values. In **Figure 4.3-7**, it shows the final bottom fiber stress with the allowable stresses. In which, *gray color* represents the *allowable value*, *top curve* represents the *tensile stress* and *bottom curve* represents the *compressive stress* for the bottom fiber. If the calculated stress is not within the limit, i.e., the top or bottom curve is outside the gray portion; you need to modify the forces to optimize the design.

The third diagram, **Post-tensioning Diagrams** shows the required and provided post-tensioning force at 1/20th points along each span. The *vertical line* represents the *required* post-tensioning and the *horizontal line* represents the *provided* post-tensioning at that section. In this *Tendon Selection* mode, provided post-tensioning force is not a straight line as in Force selection, it is varying along the length of the tendon. This variation accounts for the stress losses in the tendon due to both immediate and long-term effects. At each design section along a span, the program performs an analysis based on the post-tensioning force at that section.

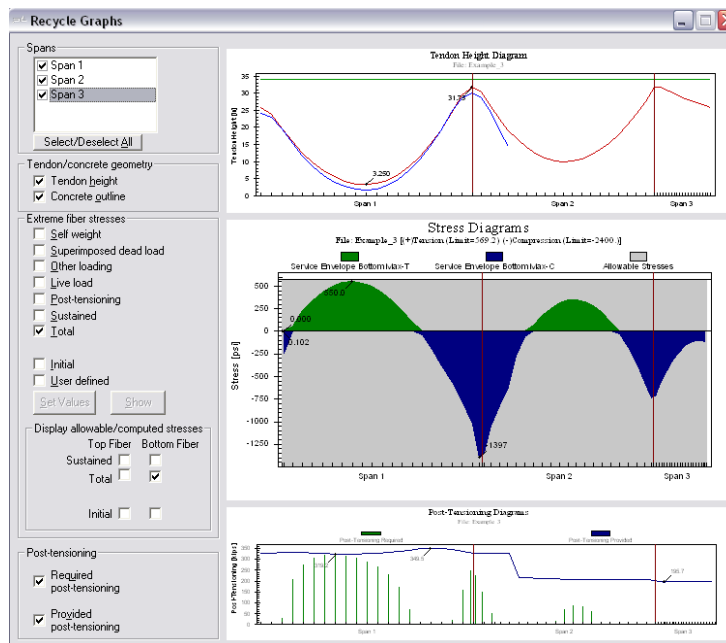


FIGURE 4.3-7

If the solutions are not acceptable, you can change the post-tensioning layout and recycle until an acceptable solution is reached. Once you are satisfied with the solution, select **Exit** at the top left of the screen to continue with the calculations.

The program continues with the calculations based on the most recent tendon forces and profile selection. Once finished successfully, you return to the main program window with the screen as shown in **Fig. 4.3-8**.

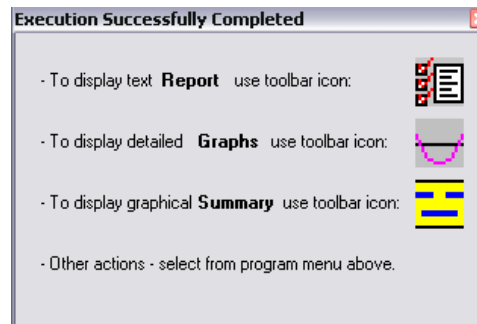



FIGURE 4.3-8

Close the above window by clicking **X** at the top right corner.

4.4 CREATE REPORTS

PT 2012 includes a Report Generator allowing the user to create full tabular, graphical reports or to customize any report according to predetermined report sections. To setup the report, select the **Report Setup** item on the *Options* menu or click the **Report Setup** button  on the main toolbar. The Report Generator screen shown in **Figure 4.4-1** will open.

The program allows you to generate reports in an MS-Word® editable format. You have the following options as explained below:

- Report cover: Select this option to generate a report cover with your logo and company information. To update your company information, click on **Update Company Info** on the *Report Generator* and you will see the screen **Company Information** shown in **Figure 4.4-2**.
- Table of Contents
- Concise Report: This report includes Project Design Parameters and Load Combinations as well as a Design Strip Report containing Geometry, Applied Loads, Design Moments, Tendon Profile, Stress check / Code check, Rebar Report, Punching Shear and Deflections. The program now reports Material Quantities in this report.
- Tabular Reports – Compact
- Tabular Reports – Detailed: This report now includes Demand Moments and Moment Capacities
- Graphical Reports
- Legend

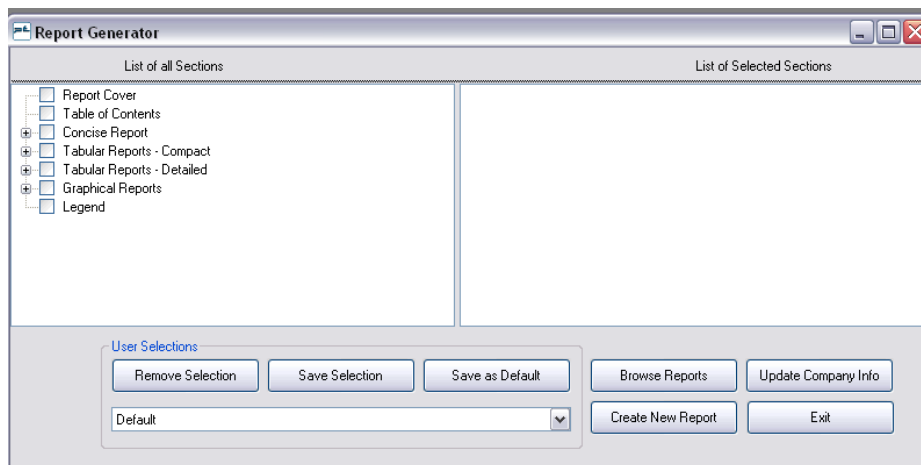


FIGURE 4.4-1

Simply check any item in the *List of all Sections* to include it in the report. The item will then appear in the *List of Selected Sections* on the right hand side of the *Report Generator*.

To generate and view the report, click on **Generate/View Report** on the bottom of the *Report Generator*.

The program allows you to open and view existing reports by clicking on **Open Reports**.

The Report Generator allows you to save report content as either a default template or as a user defined template. This enables you to quickly select content for any project by either using the default content or any other user defined content.

To define content as the default template, select report content from the List of all Sections and click on **Save as Default**.

To define content as a user defined template, select report content from the List of all Sections and click on **Save Selection**. You are asked to enter a name for your selection. This name appears then in the drop down box in the **User Selections** frame.

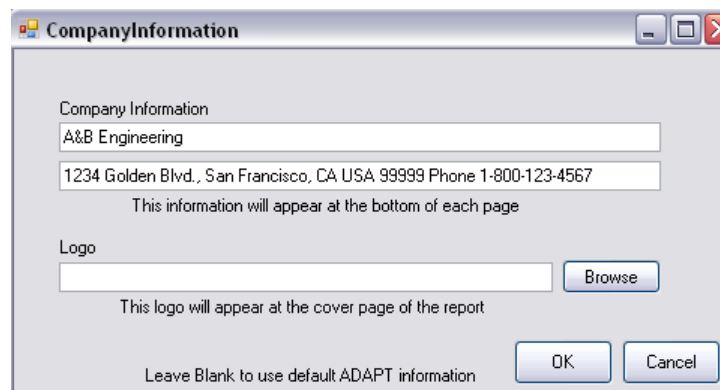



FIGURE 4.4-2

To open the “PT Summary Report” (**Fig. 4.4-3**) either click the **PTSum** button  on the tool bar or select the **PT summary** item on the *View* menu.

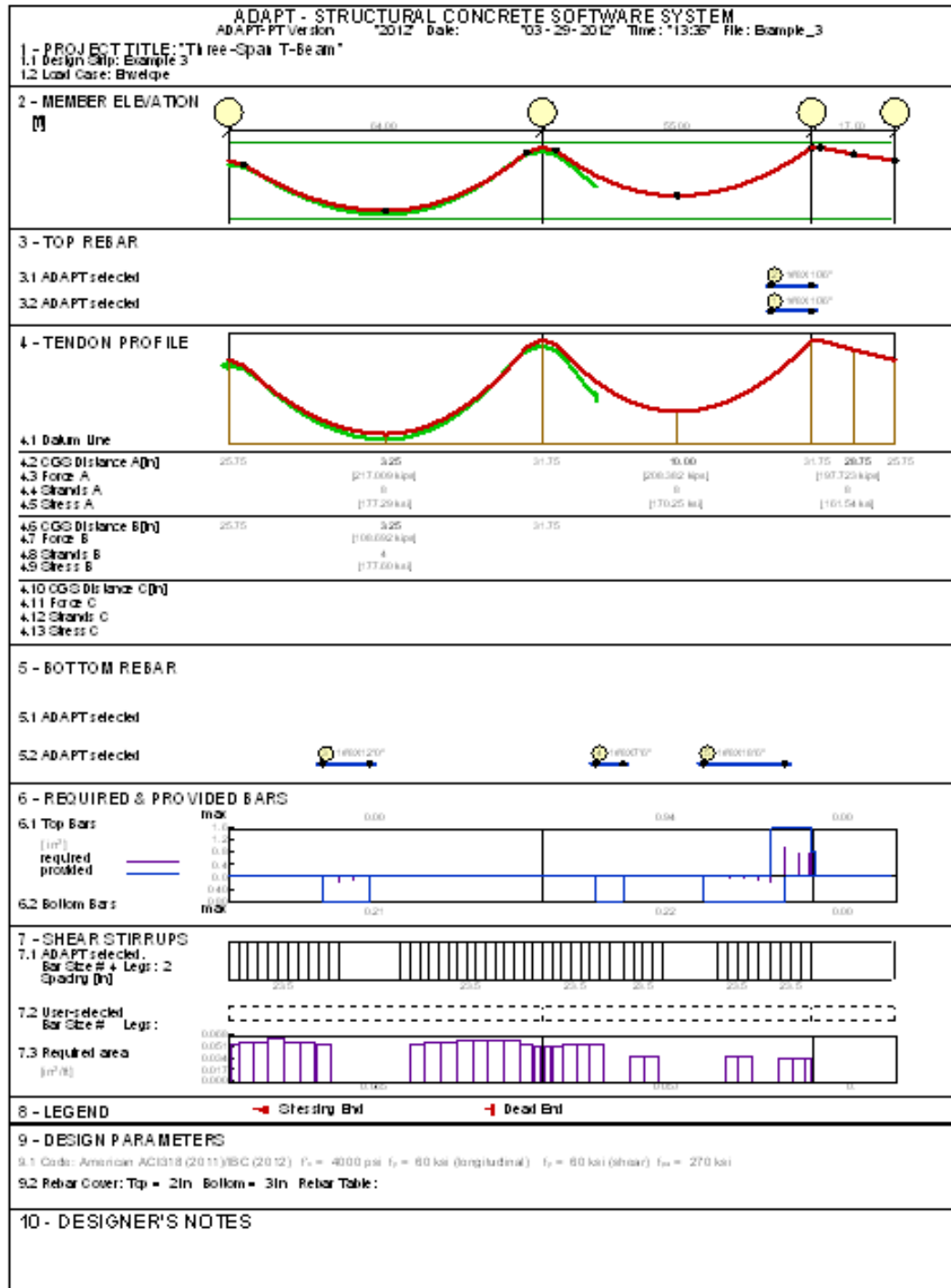


FIGURE 4.4-3

To view the graphs, either click the **Show Graphs** button  on the tool bar or select **Graphs** on the *View* menu.

5 NON-PRISMATIC (SEGMENTAL) COLUMN-SUPPORTED SLAB

Note that all new features described in the One-way Slab Example, unless noted otherwise, apply to non-prismatic (segmental) column-supported slab system models as well. Refer to Chapter 2 for new feature descriptions as they are not repeated in this chapter.

The objective of this tutorial is to demonstrate the step-by-step procedure of ADAPT-PT to model, analyze and design a three-span, non-prismatic two-way slab. Non-prismatic data entry must be used, when with the exception of drop caps and drop panels, the cross-sectional geometry of a member changes within the same span. This includes steps above or below the member, and changes in the tributary of the slab. If the change in geometry within **all** the spans of a given structure is limited to that of a drop cap and drop panel, conventional data input can be used and is more convenient for data entry. But, if in any of the spans, there is a different type of geometry change, the data input for the entire structure should use the “non-prismatic” option. This tutorial illustrates the features of data generation for non-prismatic members and walks you through a detailed example.

The procedure outlined in this tutorial is equally valid for non-prismatic one-way slabs or beams.

The geometry, material, loading and other criteria of the structure are given in the following. Views of the geometry is illustrated in **Figure 5-1**.

(i) Material Properties

5.1 Concrete:

Compressive strength, f'_c	= 4000 psi	(27.58 MPa)
Weight	= 150 pcf	(2403 kg/m ³)
Modulus of Elasticity	= 3604 ksi	(24849 MPa)
Age of Concrete at stressing	= 3 days	
Compressive strength at stressing, f'_{ci}	= 3000 psi	(20.68 MPa)

5.2 Prestressing:

Low Relaxation, Unbonded System		
Strand Diameter	= ½ in	(13 mm)
Strand Area	= 0.153 in ²	(99 mm ²)
Modulus of Elasticity	= 28000 ksi	(193054 MPa)
Ultimate strength of strand, f_{pu}	= 270 ksi	(1862MPa)
Minimum strand cover		
From top fiber	= 0.75 in all spans	(19.05 mm)
From bottom fiber		
Interior spans	= 0.75 in	(19.05 mm)
Exterior spans	= 1.5 in	(38.1 mm)

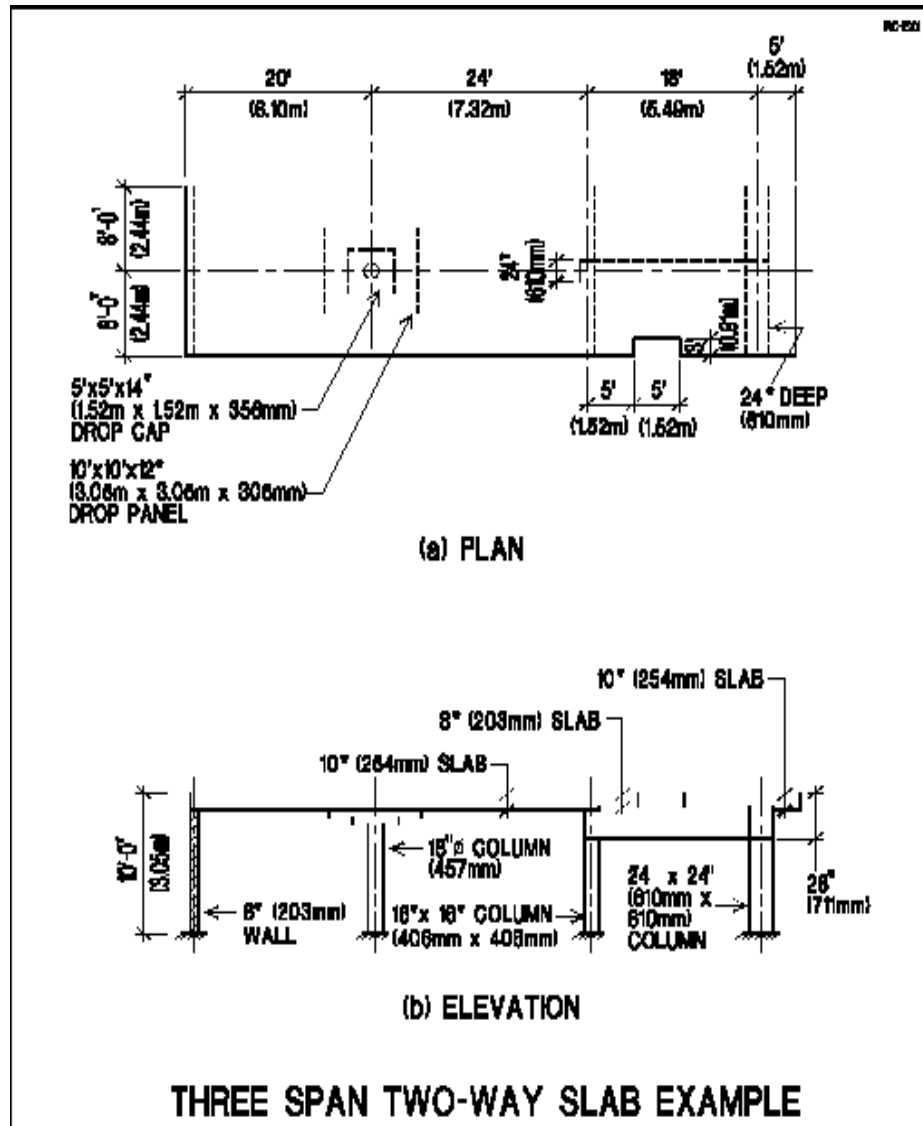


FIGURE 5-1

- Non-prestressed Reinforcement:

Yield stress f_y	= 60 ksi	(413.69 MPa)
Modulus of Elasticity	= 29000 ksi	(199,949 MPa)
Minimum Rebar Cover	= 1in Top and Bottom	(25.4 mm)

(ii) Loading:

- | | |
|-----------|------------------------------------|
| Dead load | = self weight + 20 psf |
| Live load | = 40 psf (1.92 kN/m ²) |

Since the third span of the structure is non-prismatic, i.e., the tributary width of the section changes within the span, the input of the entire structure shall utilize the “non-prismatic” option.

5.3 GENERATE THE STRUCTURAL MODEL

In the ADAPT-PT 2012 screen, click the *Options* menu and set the default code as **ACI318-2011** and the *default units* as **American**.

5.3.1 Edit the project information

5.3.1.1 General Settings (Fig. 5.3-1)

Open the new project by clicking either **New** on the *file* menu or the **New Project** on the toolbar. This automatically opens the *General Settings* input screen as in **Figure 5.1-1**. You can enter the “General Title” and the “Specific Title” of the project. For the purpose of this tutorial, enter the *General title* as **Three-Span Non-prismatic Two-Way Slab**. This will appear at the top of the first page of the output. Enter the *Specific title* as **Example 4**. This will appear at the top of each subsequent page of the output.

Next, select the *Structural System* as **Two-Way slab**. Then you will be given an option to include drop caps, transverse beams and/or drop panels. This option depends on the Geometry Input.

Next, select the *Geometry Input* as **Segmental**, since the tributary width of the third span changes along the span. Note that when you select the “Geometry Input” as “Segmental”, the option to include the drops and transverse beams is not available. Therefore, drop cap, drop panel and transverse beam in this tutorial are generated as a non-prismatic section.

Click **Next** at the bottom right of this screen, to open the *Design Settings* input screen.

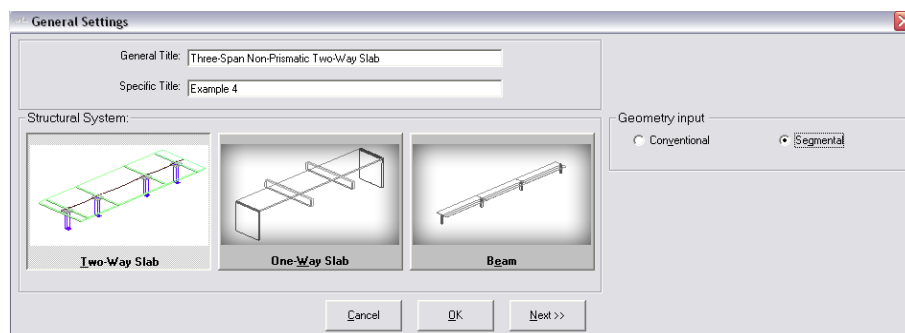


FIGURE 5.3-1

5.3.1.2 Design Code (Fig. 5.3-2)

In the second step you can specify the design code. PT 2012 now includes **ACI 318-2011/IBC 2012**

At the onset of the new model, we already selected the **ACI318-2011/IBC 2012**. This should be selected in the Design Code Screen as shown in **Fig. 5.3-2**.

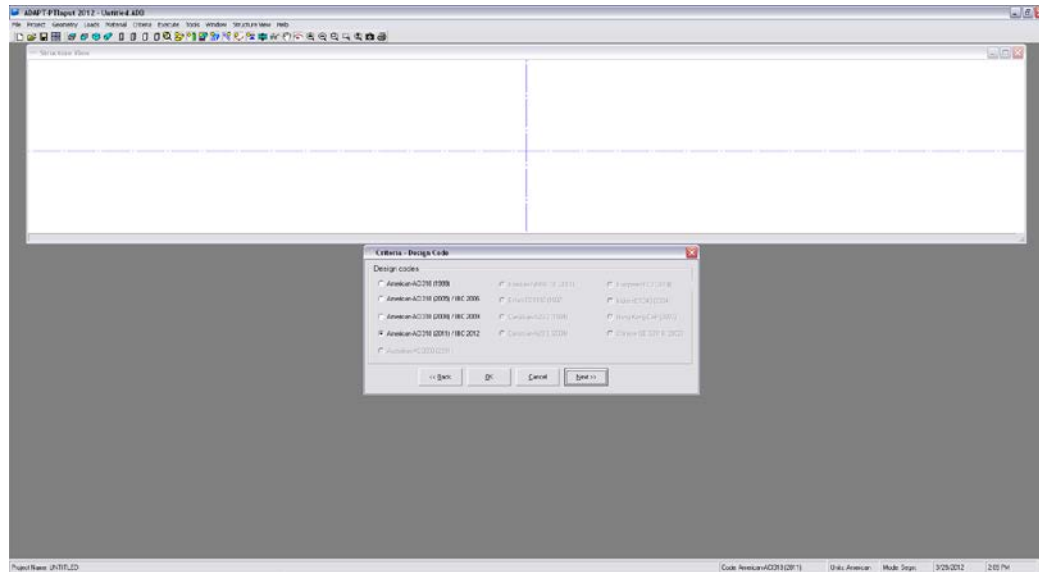


FIGURE 5.3-2

Click **Next** at the bottom right of the *Design Code* screen, to open the *Design Settings* input screen.

5.3.1.3 Design Settings (Fig. 5.3-3)

This screen is divided into three parts: *Analysis options*, *Design options*, and *Contribution to unbalanced moment*.

In *Analysis options*, you can select various calculation settings. First, select the *Execution Mode* as **Interactive**. In this mode, you have an opportunity to optimize the design by adjusting the tendon forces and tendon drapes for each span in the “Recycle” window. This will be explained later in this section.

Next, select **Yes** for *Reduce Moments to Face-of-Support* option. It indicates that the calculated centerline moments, at each support, are adjusted to the face-of support. In addition to the centerline moments, ADAPT-PT prints out the moments reduced to face-of- support.

Select **No** for the option to *Redistribute moments*.

For a two-way slab system, you have an option of modeling the structure using the *Equivalent Frame* method. If it is not used, there is an option to *Increase Moment of Inertia Over Support*. This option will cause the program to use a larger moment of inertia over the supports than given by the cross-sectional geometry of the beam.. This, in turn, affects the relative distribution of the moments and may affect the amount of post-tensioning required. Select **No** for both *Equivalent Frame Modeling* and *Increase Moment of Inertia Over Support*.

In *Design options*, you can either *Use all provisions of the code* that you selected in the previous step, or *Disregard the following provisions* such as *Minimum rebar for serviceability*, *Design capacity exceeding cracking moment*, and *Contribution of prestressing in strength check*.

Next select the option for *Generate moment capacity based on* **Design Values** or **User-Entered Values**. If **Design Values** is selected, the program will calculate and report positive and negative moment capacities based on prestressing steel, base reinforcement as defined by the user (this is discussed later in this section) and program-calculated reinforcement. Demand moments at 20th points along each span are also reported. When **User-Entered Values** is selected, the program will calculate and report similar moment capacities and demand moments where the capacities are based on prestressing steel and based reinforcement as defined by the user. For this example, select **Design Values**.

In *Contribution to unbalanced moment*, you specify the contribution of either *Top isolated bars*, and *Bottom isolated bars*, and *Post-tensioning* in percent. Leave the default values (100%).

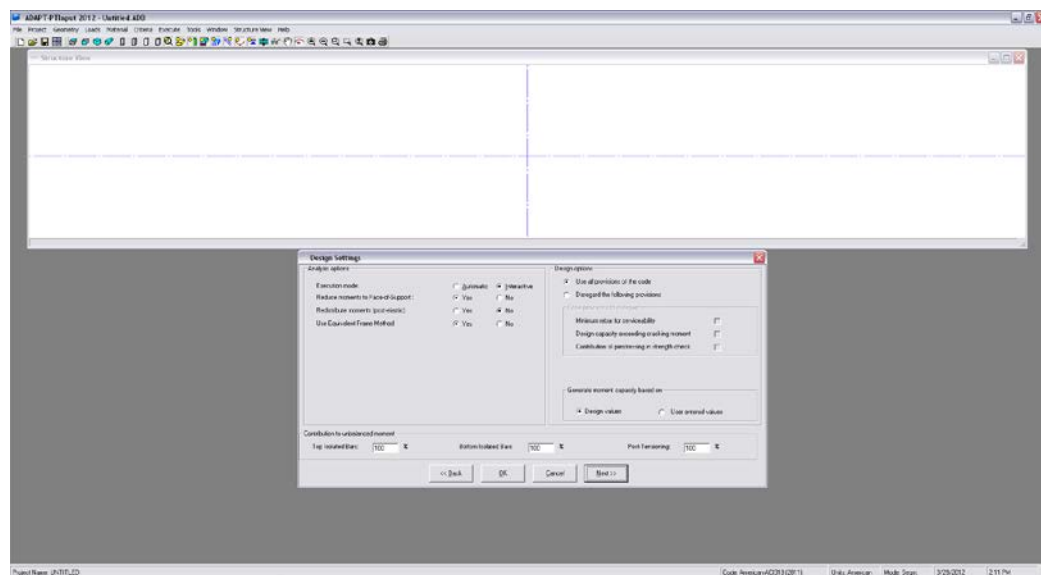


FIGURE 5.3-3

Click **Next** at the bottom right of the *Design Settings* screen, to open the *Span Geometry* input screen.

5.3.2 Edit the geometry of the structure

5.3.2.1 Enter Span Geometry (Fig. 5.3-4 -14)

This screen is used to enter the cross-sectional geometry of the slab.

As you enter the values, the span is displayed in real-time in the 3D window. You can zoom in and out in the *Structure View* with the help of your mouse wheel or with the help of the *Zoom In* or *Zoom Out* buttons in the *View Toolbar*.

You can access special data editing options by selecting data cells and right clicking. Available options include Insert New Line, Delete Line, Copy Selected Lines, and Paste Lines.

Set the *Number of Spans* to **3** either by clicking the **up arrow** or using **CTRL +**. Then click on the checkbox next to “R-Cant” in the *Label* column to include the right cantilever.

Next, enter the dimensions. All dimensions are defined in the legend at the top of the screen and/or illustrated in the appropriate section figure. The section type for any span can be changed by clicking on the button in the section (*Sec*) column.

Since the structure is non-prismatic, you have to enter the data for all the segments in each span. **Figure 5.3-4** and **Figure 5.3-5** are the 3-D and top perspective views of the model showing the subdivision of the system based on change in its cross-sectional geometry. The structure includes 8” (203 mm) wall at the first support, 18” (457 mm) diameter circular column with drop cap and drop panel at the second support, 16” (406 mm) square column with a longitudinal beam at the third support, 24” (610 mm) square column with a transverse beam at the fourth support and 5’ (1.52 m) cantilever beyond the last span. The cross-section changes at the face of the drop cap and drop panel, and at the face of the transverse beam. Also the tributary width changes in the third span. You have to enter the data for each segment individually.

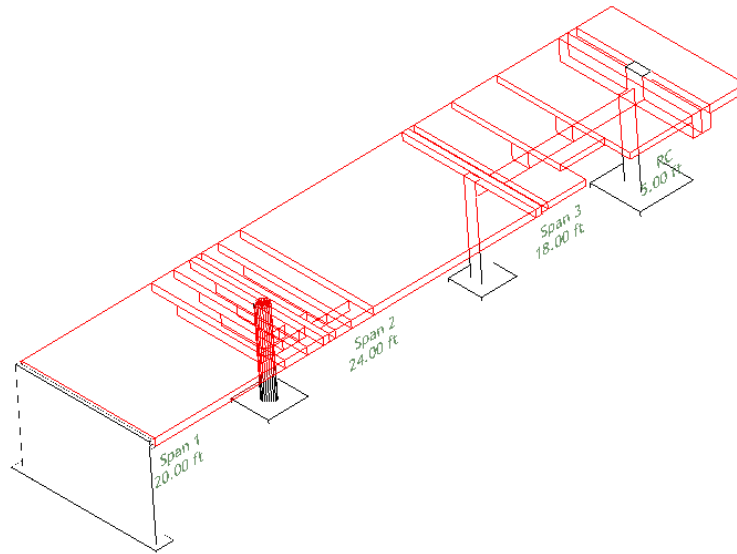


FIGURE 5.3-4

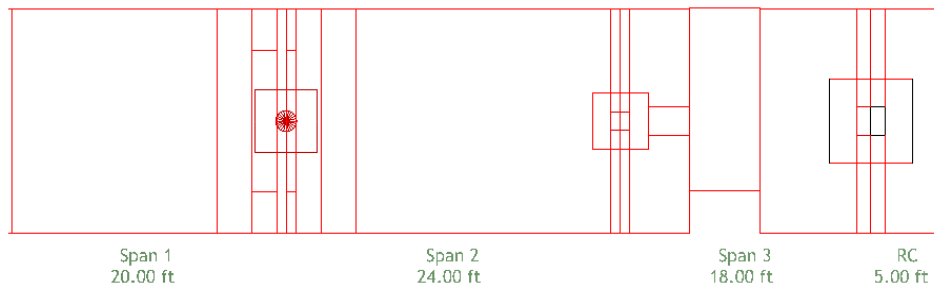


FIGURE 5.3-5

In the *Span Geometry* input screen (**Fig. 5.3-6**) enter the cross-sectional geometry of each span at its mid-length (mid-span, *Sec*, *L*, *b*, *h* etc), as if the entire span were prismatic. Input data for the remainder of the segments of each span will be entered later in a second input screen. For the first span, select the *Sec* as **rectangular**, edit *L* as **20** ft (6.10 m), *b* as **192** inches (4877 mm) and *h* as **10** inches (254 mm) . Repeat this procedure for rest of the spans as shown in **Figure 5.3-6**. To enter the cantilever dimensions, click with your mouse on the grey check box on the left next to **R-Cant** in the *Label* column. A red check mark will be added to the grey check box.

To enter the data for all the segments in each span, change *PR* (Prismatic) column to **NP** (Non-prismatic) from the drop down list for all the spans. This activates the **More...** button in the Segments (*Seg*) column (**Fig. 5.3-6**).

Label	PR	Sec	Seq	L	b	h	bf	hf	b _m	h _m	R _h	<- M =	M -> =
Typical	PR			0.00	0.00	0.00					10.00	0.50	0.50
L-Cant	PR			20.00	192.00	10.00					10.00	0.50	0.50
SPAN 1	PR			24.00	192.00	10.00					10.00	0.50	0.50
SPAN 2	PR			18.00	24.00	28.00	156.00	8.00			10.00	0.62	0.38
SPAN 3	PR			5.00	192.00	10.00					10.00	0.50	0.50
R-Cant	PR			5.00	192.00	10.00					10.00	0.50	0.50

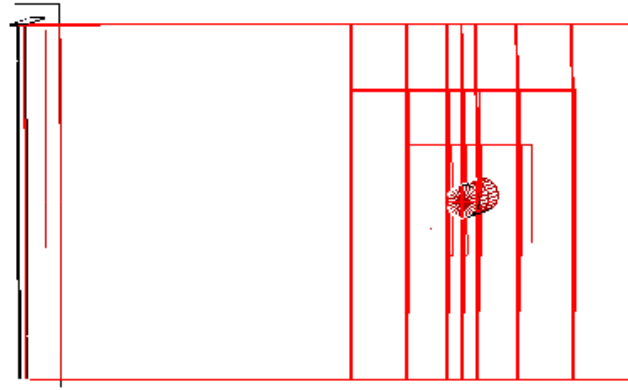
FIGURE 5.3-6

Clicking on the **More...** button opens the *Geometry-Span (More)* window for that span. This is where you enter the cross-sectional geometry of the remainder of the segments of each span.

Click the **More...** button in Span1 to enter the segmental data for that span (**Fig. 5.3-8**). **Figure 5.3-7** is the perspective view of the first span, which shows the locations where cross -sections change within that span. There are five segments in the first span. The list below shows each segment and its location:

- Segment 1** – Automatically created by the program and located at the centerline of wall.
- Segment 2** – Automatically created by the program and located at the face of the first support.
- Segment 3** – Represents the flanged section at the beginning of the drop panel at the first interior support (Support 2).
- Segment 4** – Represents the extended ‘T’ section where the drop cap begins at the first interior support (Support 2).
- Segment 5** – Automatically created by the program and is a copy of Segment 4 and extends to centerline of the first interior support (Support 2).

Set the *Number of Segments* as **5**. Up to seven segments may be entered per span.

**FIGURE 5.3-7**

The parameters are input in the same manner as the mid-span span geometry, except for the *XL* column, which is used to specify the distance from the left support centerline to the start of each segment. The length of each segment is calculated automatically based on the distance to the start of the next segment.

Note 1:

*The **XL** column of the **More....** input screen will differ according to the selection in **design settings** input screen. The following flow chart summarizes the procedure.*

*If you select either **Equivalent Frame** or **Increase Moment of Inertia** in **Design Settings** screen, the **More...** input screen for the first span would be as shown in the following figure. In this case, the program automatically generates additional segments over each support using the geometry entered for the first and last segments. If these segments are generated before the support dimensions are entered, their *XL* values will be initialized with values of zero and the span length, respectively. These values will be updated when the support dimensions are entered.*

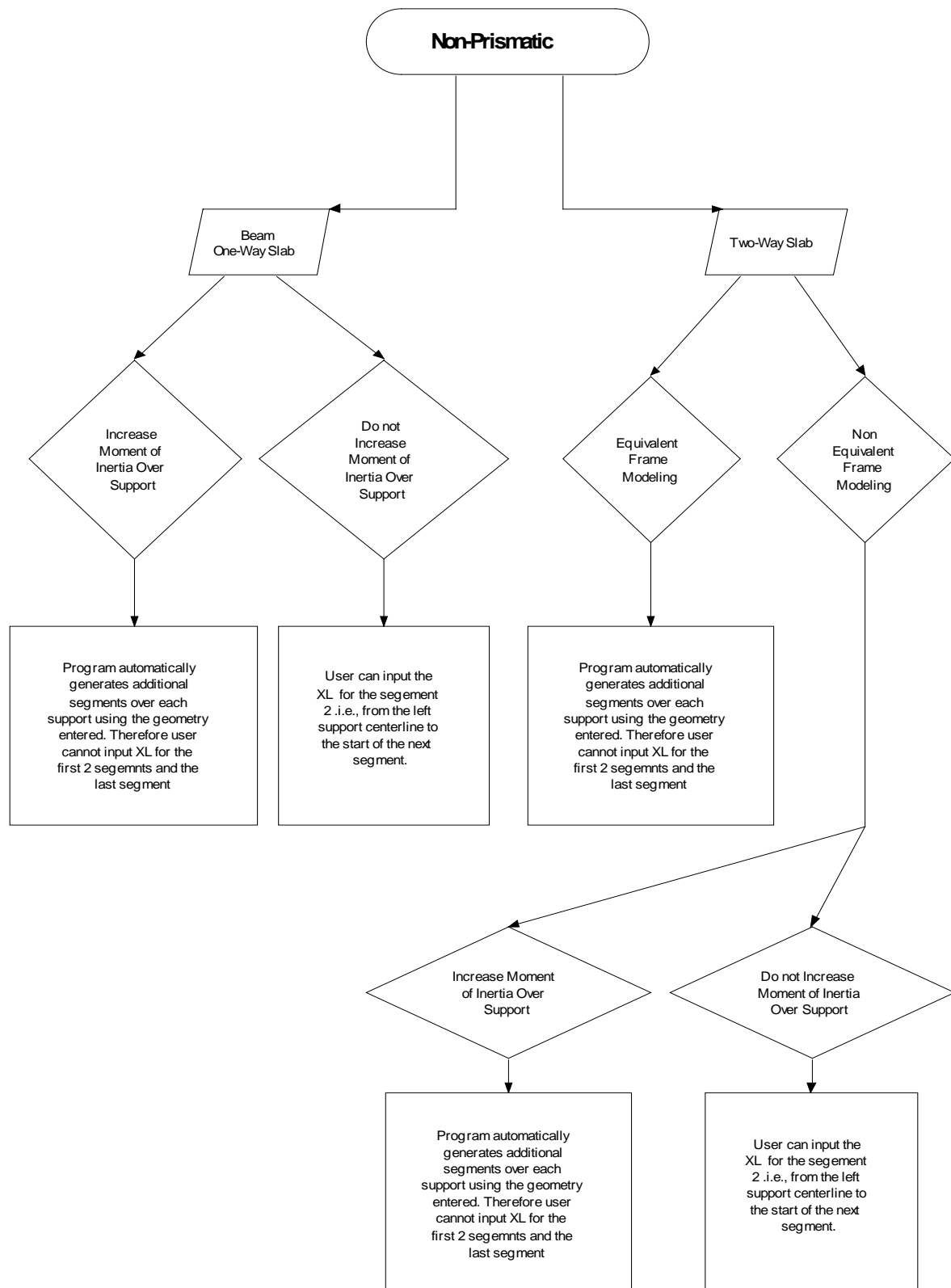


FIGURE NOTE 1-2

The first and last segments are always reserved for the portion of the slab (or beam) that falls over the support. These are calculated automatically by the program. The second segment will also be generated automatically by the program. The segment distance, XL , in this case will be equal to $\frac{1}{2}$ the support dimension. You start by entering the geometry of the third segment. The start of the first segment is always zero. Enter XL for *Segment 3* as **15 ft (4.57 m)** (distance from the left support centerline to the face of the drop panel) and *Segment 4* as **17.5 ft (5.33 m)** (distance from the left support centerline to the face of the drop cap). Note that you cannot modify the XL column for the first, second and last segment.

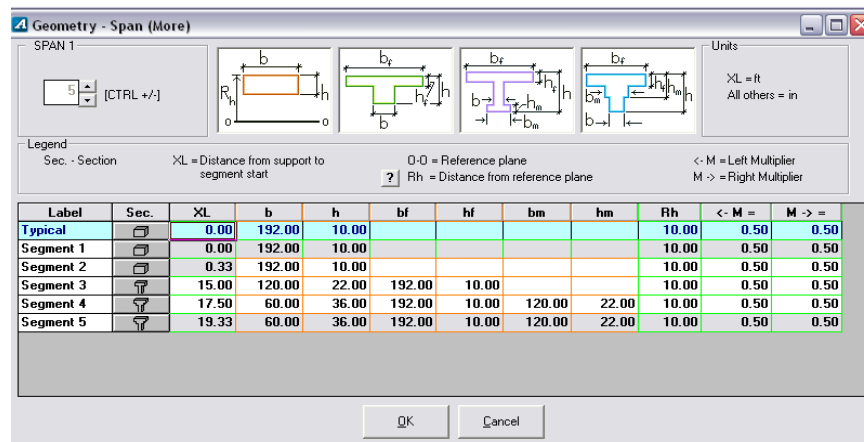


FIGURE 5.3-8

Select the *section* for segments 1 and 2(0-15 ft) as **rectangular** with b equal to **192 inches (4879 mm)** (tributary width) and h equal to **10 inches (254 mm)**.

For segment 3 (15 – 17.50 ft, face of drop panel to the face of drop cap) select *section* as **T** and enter **120 inches (3048 mm)** (width of the drop panel) for b , **22 inches (559 mm)** (depth of the drop panel + thickness of the slab) for h , **192 inches (4877 mm)** (tributary width) for bf , **10 inches (254 mm)** (thickness of the slab) for hf .

For segment 4 (spanning 17.50-20 ft, i.e., from the face of the drop cap to the centerline of the second support), select *section* as **Extended T Section** and enter **60 inches (1524 mm)** (width of the drop cap) for b , **36 inches (914 mm)** (depth of the drop cap + depth of the drop panel + thickness of the slab) for h , **192 inches (4877 mm)** (tributary width) for bf , **10 inches (254 mm)** (thickness of the slab) for hf , **120 inches (3048 mm)** (width of the drop panel) for bm and **22 inches (559 mm)** (depth of the drop panel + thickness of the slab) for hm .

Repeat the same procedure for span 2, span 3 and the right cantilever.

Span 2 also has five segments as in Span 1 (**Fig. 5.3-9**).

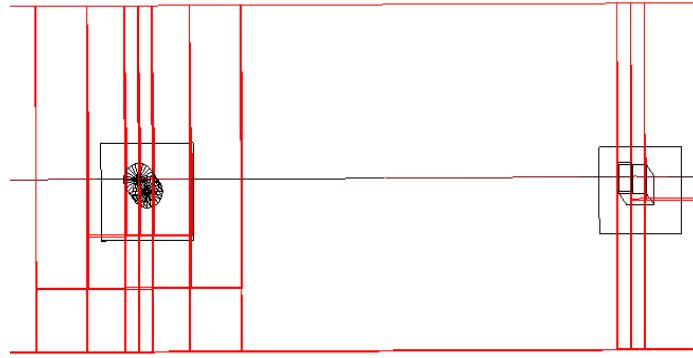


FIGURE 5.3-9

Enter the data for each segment as in **Figure 5.3-10**.

Label	Sec.	XL	b	h	bf	hf	bm	hm	Rh	< M =	M > =
Typical		0.00	192.00	10.00	10.00	10.00	10.00	10.00	10.00	0.50	0.50
Segment 1		0.00	60.00	36.00	192.00	10.00	120.00	22.00	10.00	0.50	0.50
Segment 2		0.67	60.00	36.00	192.00	10.00	120.00	22.00	10.00	0.50	0.50
Segment 3		2.50	120.00	22.00	192.00	10.00	10.00	10.00	10.00	0.50	0.50
Segment 4		5.00	192.00	10.00	10.00	10.00	10.00	10.00	10.00	0.50	0.50
Segment 5		23.33	192.00	10.00	10.00	10.00	10.00	10.00	10.00	0.50	0.50

FIGURE 5.3-10

For span 3, there is a cutout in the slab from 5 to 10 ft (1.52-3.05 m), and a transverse beam at the right support. There are 6 segments. The list below shows each segment and its location:

- Segment 1** – Automatically created by the program and located at the centerline of column (Support 3).
- Segment 2** – Automatically created by the program and located at the face of the third support.
- Segment 3** – Represents the flanged section at the beginning of the slab cutout.
- Segment 4** – Represents the flanged section at the cutout.
- Segment 5** – Represents the transverse beam section at the end of the slab cutout.
- Segment 6** – Automatically created by the program and is a copy of Segment 5 and extends to the centerline of Support 4.

Figure 5.3-11 shows a top view of Span 3 and the right cantilever.

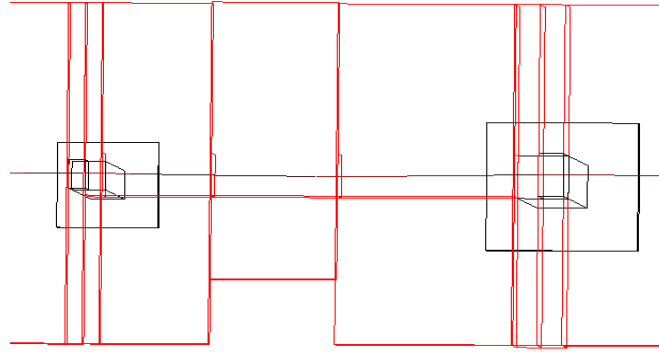


FIGURE 5.3-11

Enter the data for each segment of the third span as shown in the input screen below (Fig. 5.3-12).

Geometry - Span (More)

SPAN 3

Legend
 Sec. - Section
 XL = Distance from support to segment start
 0-0 = Reference plane
 Rh = Distance from reference plane
 < - M = Left Multiplier
 M -> = Right Multiplier

Label	Sec.	XL	b	h	bf	hf	bm	hm	Rh	< - M =	M -> =
Typical		0.00	24.00	28.00	156.00	8.00			10.00	0.62	0.38
Segment 1		0.00	24.00	28.00	192.00	8.00			10.00	0.50	0.50
Segment 2		0.67	24.00	28.00	192.00	8.00			10.00	0.50	0.50
Segment 3		5.00	24.00	28.00	156.00	8.00			10.00	0.62	0.38
Segment 4		10.00	24.00	28.00	192.00	8.00			10.00	0.50	0.50
Segment 5		17.00	192.00	28.00					10.00	0.50	0.50
Segment 6		17.00	192.00	28.00					10.00	0.50	0.50

OK Cancel

FIGURE 5.3-12

The right cantilever has two segments; centerline of the column to the face of the transverse beam, 0-1 ft (0 – 0.30 m), and the face of the transverse beam to the end of the span, 1-5 ft (0.30 – 1.52 m), as shown in Figure 5.3-11. Enter the data for the segments as shown in Figure 5.3-13.

Geometry - Span (More)

Right Cantilever

Legend
 Sec. - Section
 XL = Distance from support to segment start
 0-0 = Reference plane
 Rh = Distance from reference plane
 < - M = Left Multiplier
 M -> = Right Multiplier

Label	Sec.	XL	b	h	bf	hf	bm	hm	Rh	< - M =	M -> =
Typical		0.00	192.00	10.00					10.00	0.50	0.50
Segment 1		0.00	192.00	28.00					10.00	0.50	0.50
Segment 2		1.00	192.00	10.00					10.00	0.50	0.50

OK Cancel

FIGURE 5.3-13

Next enter the reference height. The reference height (Rh) identifies the position of a reference line that is used to specify the location of the tendon. Typically, the reference line is selected to be the soffit of the member. Hence for this tutorial, select slab depth. Click ? with the **Rh** definition in the legend for additional information regarding Reference Height.

Type the reference height, Rh , as **10** inches (254 mm) for all the segments of all spans.

The Left and Right Multiplier columns ($<-M$ and $M->$) are used to specify the tributary width to indicate how much of the tributary width falls on either side of the frame line. Enter **0.5** for both the *left* and *right* multiplier for all the segments of all spans, except for the second segment of span 3. There is a 3 ft (0.91 m) cutout, so the tributary width on either side of the frame line is different. Enter **0.62** for the *left* multiplier and **0.38** for the *right* multiplier as shown in **Figure 5.3-6**. Note that the sum of left and right multipliers add up to be 1.

You can use the “Typical” row (top row) if several spans have similar dimensions. To enter typical values, type the **value** into the appropriate cell in the top row and then press **Enter**. The typical value will be copied to all the spans.

Your structure should look like shown in **Figure 5.3-14**.

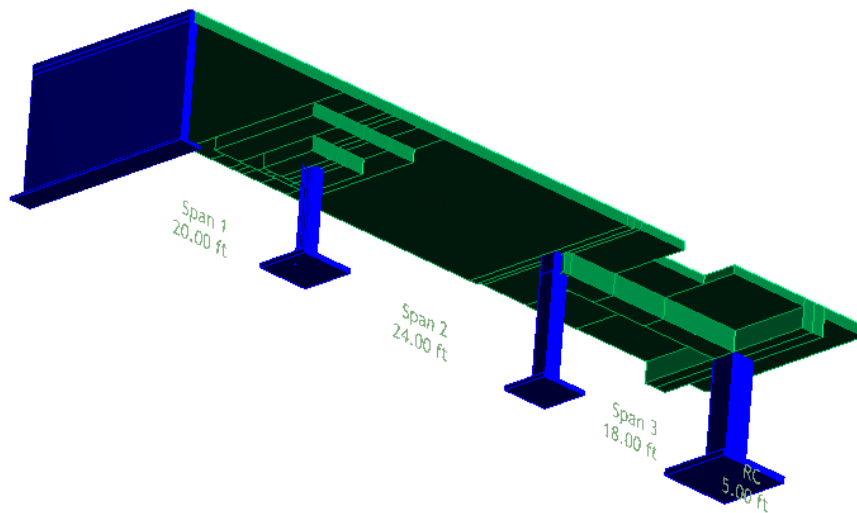


FIGURE 5.3-14

Click **Next** on the bottom line to open the input screen, *Support Geometry*.

5.3.2.2 Enter Supports-Geometry (Fig. 5.3-15)

This screen is used to input column /wall heights, widths and depths. You may enter dimensions for columns/walls above and/or below the slab.

Select **Lower column** from the *Support selection* box and enter **10 ft (3.05 m)** for *H1* in the “Typical” row (top row). Press **ENTER** to assign this value to all the lower columns.

Next, enter the dimensions of the supports. *B* is the dimension of the column cross- section normal to the direction of the frame. *D* is the column dimension parallel to the frame. For Support 1, edit the wall dimensions, *B* as **192 inches (4877 mm)** and *D* as **8 inches (203 mm)**. For other supports, enter the given column dimensions as shown in **Figure 5.3-15**.

On this input screen, you can select for each support whether the left edge and the right edge of that support is interior or exterior. If the view of slab is shown as a Plan view, the left edge is that at the top of slab and the right edge is that at the bottom of slab. When **Exterior** is selected, the program automatically checks the distance from face of column to either edge. If this distance is less than $4 \times \text{slab thickness, } h$, the program will consider the column an exterior condition (i.e. edge condition). If **Interior** is selected, the column will be treated as an interior condition. For this example, select Exterior for the Left and Right Edge.

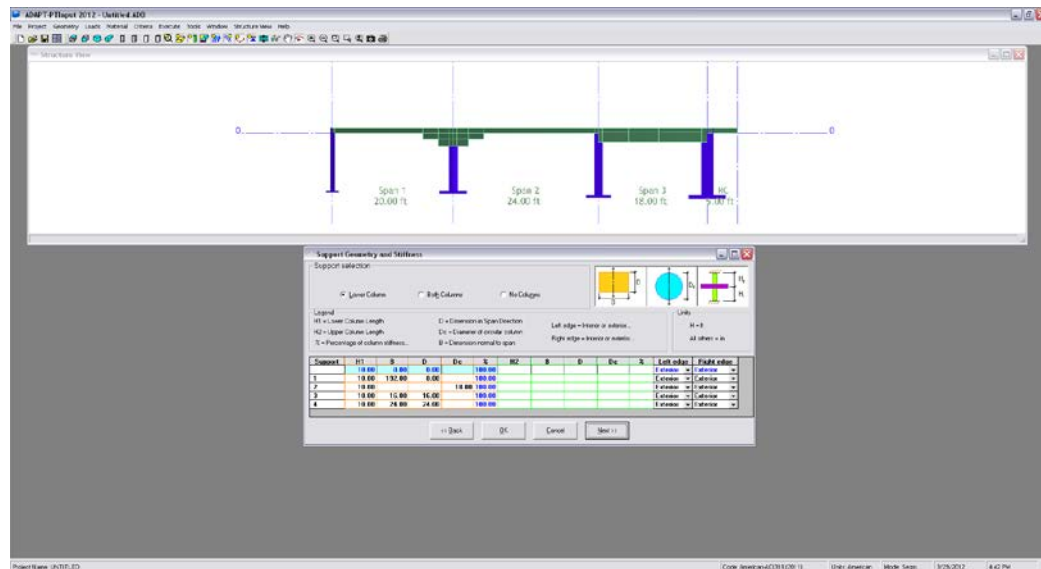


FIGURE 5.3-15

Click **Next** on the bottom line to open the next input screen, *Supports Boundary conditions*.

5.3.2.3 Enter Supports Boundary Conditions (Fig. 5.3-16)

This screen is used to enter support widths and column boundary conditions. The value you enter for “support width” in this screen is used by the program to reduce the moment to face-of-support.

Support widths can be entered if you answered “Yes” to the “Reduce Moments to face-of- support” question on the *Design Settings* screen, i.e., if you answered “No”, you cannot input values in the *SW* column. This input value will be used to calculate the reduced moments.

Since the support width, *SW*, is set to the column dimension (*D*) as a default, the *SW* values will be automatically determined from the support geometry and cannot be modified by the user. If you want to input the *SW* values, **uncheck** the *SW=Column Dimension* box.

Select *LC (N)* (Lower Column, Near end) and *LC (F)* as **1**, fixed, from the drop down list.

Leave the *End Support Fixity* as default **No**. This will be used when the slab or beam is attached to a stiff member.

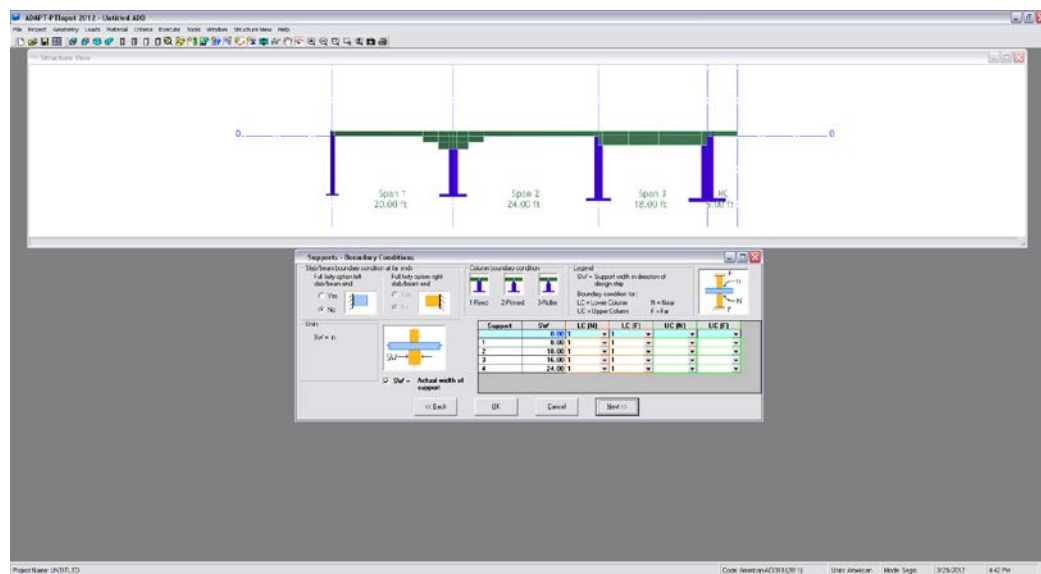


FIGURE 5.3-16

Click **Next** at the bottom of the screen to open the next input screen. *Loading*.

5.3.3 Enter Data

5.3.3.1 Edit the loading information (Fig. 5.3-17)

Any number of different loads and load types may be entered for a span.

Load types available in PT 2012 are: *Uniform, Partial Uniform, Concentrated, Moment (Concentrated), Line, Triangular, Variable and Trapezoidal*.

Enter the span number as **1** in the *Span* column. If the loads are the same for all the spans, you can type **ALL** or **all** in the *Span* column. This will copy the data to all the spans.

If you choose not to include Self-weight, you now have the option to define the self-weight (**SW**) as a *Class*. In any case, you can choose to specify additional dead load as Superimposed Dead Load (**SDL**) as a *Class*. PT 2010 gives you the option to specify any load as an **X Class**.

Select the *Class* as **SDL** from the drop down list and specify the load type as uniform either by typing **U** in *L-?* or by **dragging the icon** from the graphics of the uniform loading. The default of the load type when you select the load *Class* is **L-U**; so leave it as is for this tutorial.

Type **0.02 k/ft²** (0.96 kN/m²) (without self-weight) for dead load in the *w* column. You can enter DL with or without self-weight, since the program can calculate self-weight automatically. In order to be calculated automatically, you must answer **Yes** to the *Include Self-Weight* question at the top right of the screen and also must enter a unit weight of concrete. Type **150 pcf** (2402.85 kg/m³) as the *Unit Weight*.

Repeat the procedure for live load by entering the **span number** and changing the *Class* to **LL** and the *w* value to **0.04 k/ft²** (1.92 kN/m²) for all the spans.

Answer **Yes** to *Skip Live Load?* at the top left of the screen and enter the *Skip Factor* as **1**.

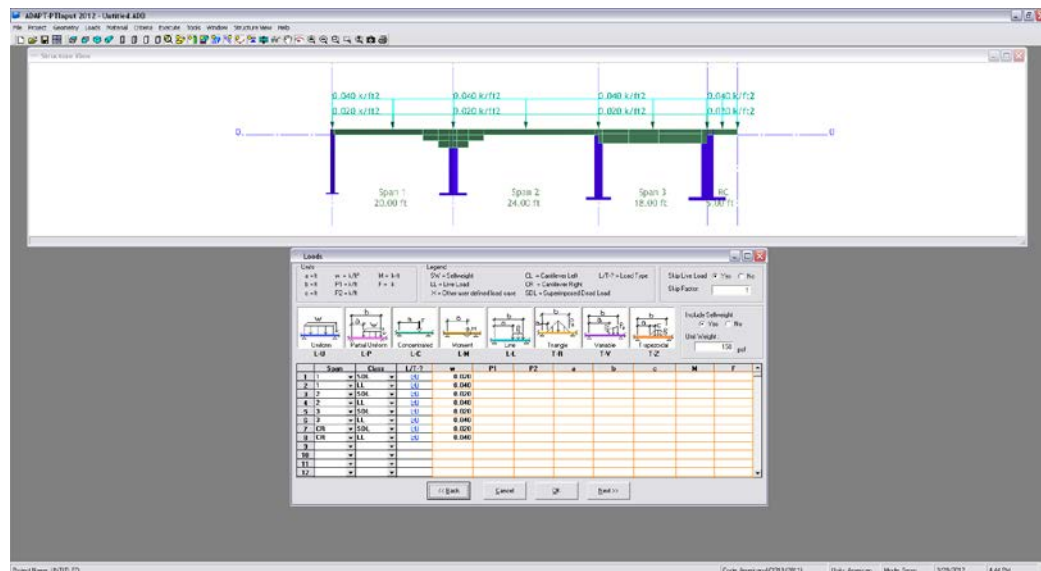


FIGURE 5.3-17

Click **Next** at the bottom of the screen to open the next input screen, *Material-Concrete*.

If you entered span as “all”, click **Back** and go back to the loading screen. You can see that all the loads are copied to the individual spans as in **Figure 5.3-17**.

5.3.4 Edit the material properties

5.3.4.1 Enter The Properties Of Concrete (Fig. 5.3-18)

Select the **Normal weight** and **enter** the *strength at 28 days* for slab/beam and column. When you press **enter** from the strength input value, the *Modulus of Elasticity* will be calculated automatically based on the concrete strength and the appropriate code formula. For this tutorial, keep the default values for concrete strength and creep coefficient. The creep coefficient will be used in the calculation of long-term deflection.

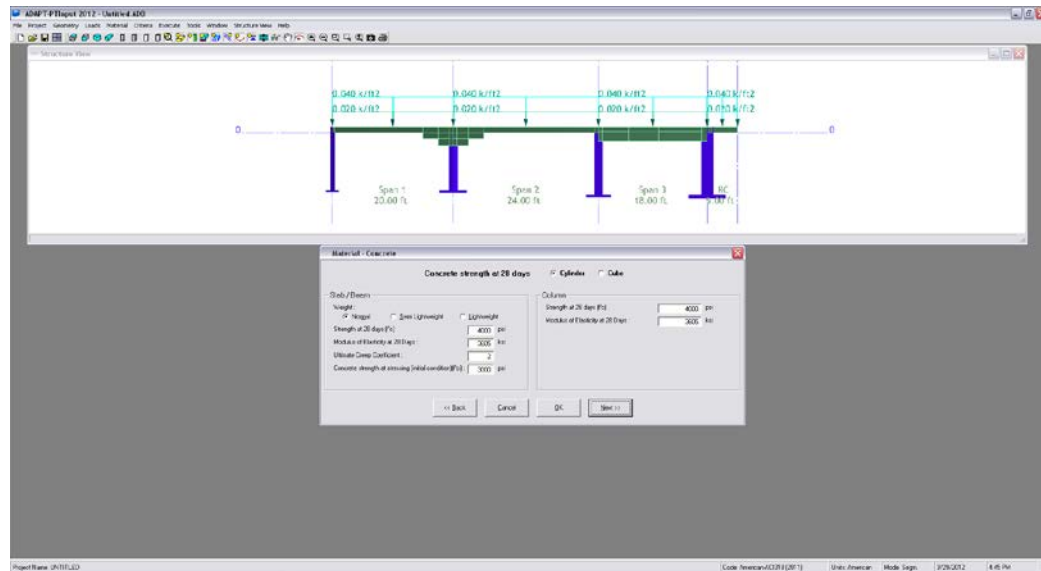


FIGURE 5.3-18

Click **Next** at the bottom of the screen to open the *Material Reinforcement* input screen.

5.3.4.2 Enter The Properties Of Reinforcement (Fig. 5.3-19)

The screen is divided into two parts: *Longitudinal reinforcement* and *Shear reinforcement*.

Edit the properties of reinforcement and bar sizes. Change the bar sizes according to **Figure 5.3-19**. The preferred bar sizes for top and bottom will be used when calculating the number of bars required.

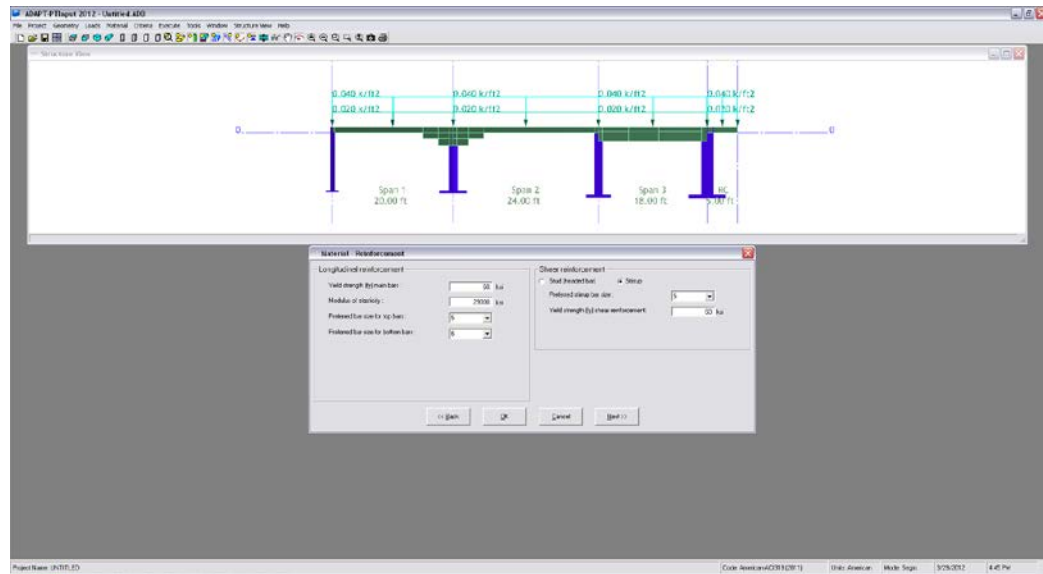


FIGURE 5.3-19

Click **Next** at the bottom of the screen to open the next input screen.

5.3.4.3 Enter The post-tensioning system parameters (Fig. 5.3-20)

Select the *Post-tensioning system* as **Unbonded** and use the default values for the other defined parameters.

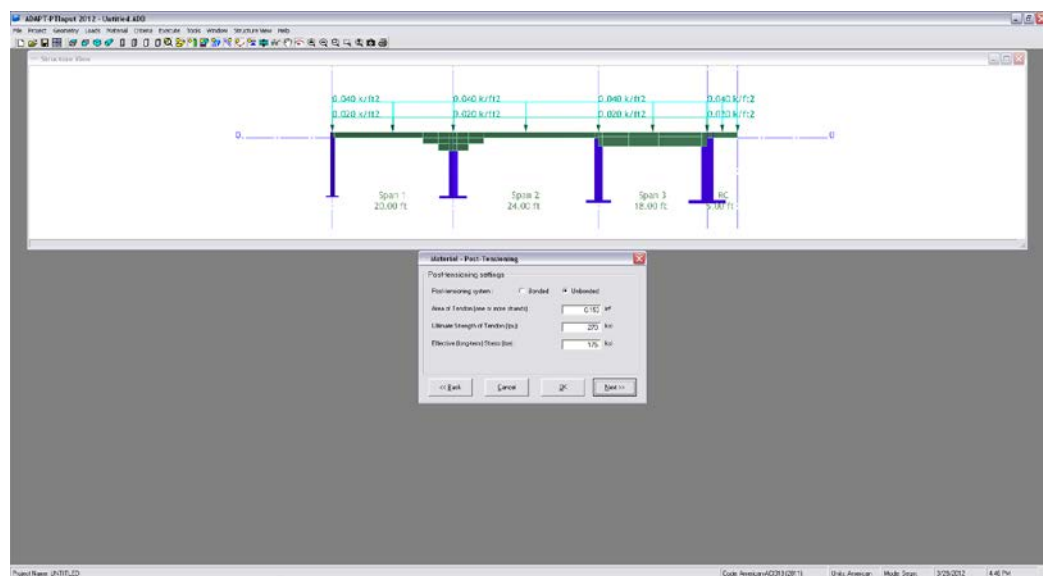


FIGURE 5.3-20

Click **Next** at the bottom of the screen to open the next input screen, *Base Non-Prestressed Reinforcement*.

5.3.4.4 Edit Base Reinforcement (Fig. 5.3-21)

The program allows you to specify a base reinforcement that is taken into consideration when designing the structure. The base reinforcement can be input as mesh or isolated rebar. For this example, base reinforcement will not be considered. Select **No** in the *Base Reinforcement* section.

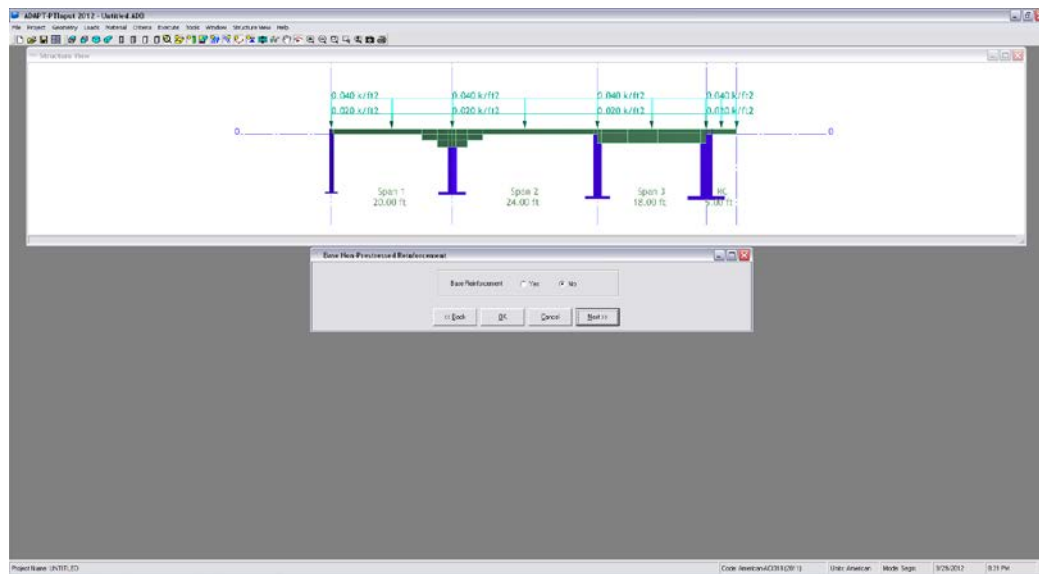


FIGURE 5.3-21

Click **Next** at the bottom of the screen to open the next input screen, *Criteria Allowable Stresses*.

5.3.5 Edit the design criteria

5.3.5.1 Enter The Initial And Final Allowable Stresses (Fig. 5.3-22)

Tensile stresses are input as a multiple of the square root of f'_c , and compressive stresses are input as multiple of f'_c .

Change the top and bottom final tensile stress to $6\sqrt{f'_c}$ ($0.75\sqrt{f'_c}$). This is the allowable tensile stress per ACI 318-2011 for a two-way prestressed slab.

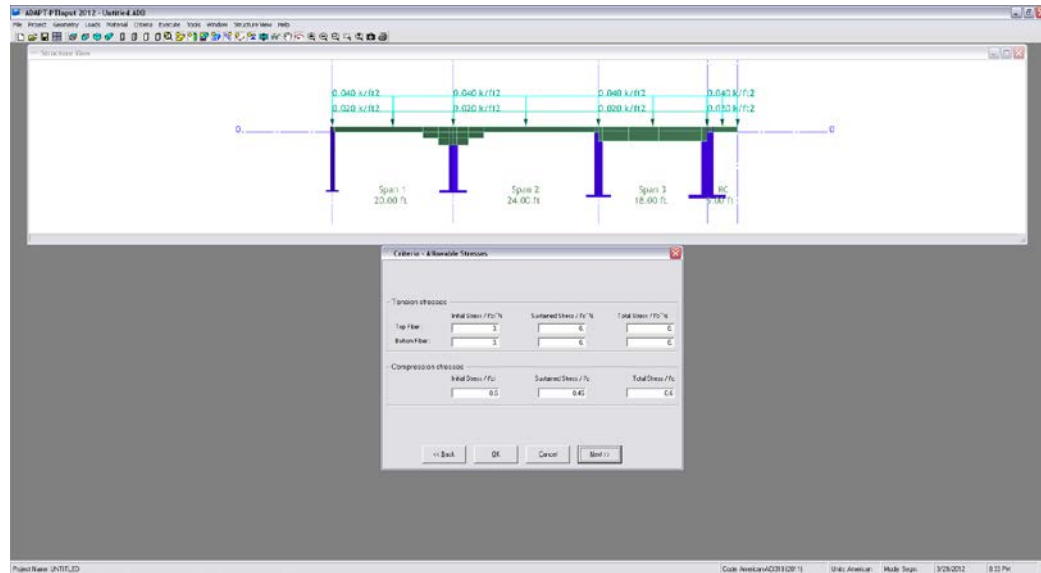


FIGURE 5.3-22

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Recommended Post-Tensioning Values*.

5.3.5.2 Enter The Recommended Post-Tensioning Values (Fig. 5.3-23)

This screen is used to specify minimum and maximum values for average precompression (P/A : total prestressing divided by gross cross-sectional area) and percentage of dead load to balance (W_{bal}). These values are used by the program to determine the post-tensioning requirements and the status of the P_{min}/P_{max} and W_{BAL} Min/ Max indicators on the “Recycle” window.

The values given as default are according to the code and the experience of economical design. Keep the **default values**.

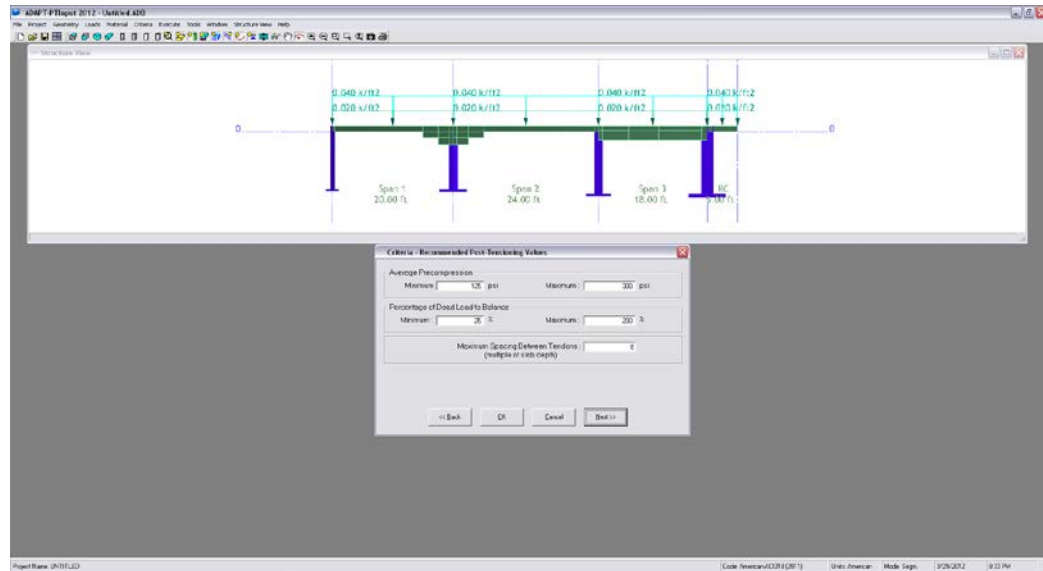


FIGURE 5.3-23

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Calculation Options*.

5.3.5.3 Select The Post-Tensioning Design Option (Fig. 5.3-24)

The two design options are “Force Selection” and “Force/Tendon Selection”, as in **Figure 5.3-24**. **Force Selection** is the default option and will be used for this example.

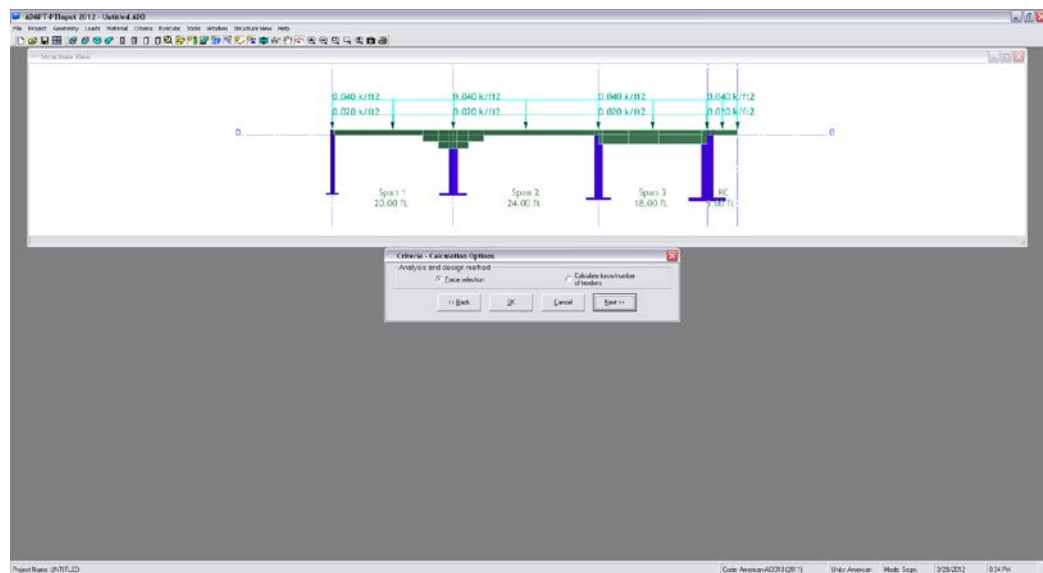


FIGURE 5.3-24

In this option, a tendon will be assigned a final and constant effective force, equal to the jacking force minus all stress losses, expressed as a single value.

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Tendon Profile*.

5.3.5.4 Specify The Tendon Profiles (Fig. 5.3-25)

The program allows you to specify up to three tendon paths per span. You can define one profile for each of the three tendons.

In the section *Option for tendons* you can define the *Default extension of terminated tendon as fraction of span*. Also, you can specify the *Shape of tendon extension* from the *Left end* and the *Right end*. For this example, leave the default values.

In this example we only use tendon A. Select **1** (Reversed parabola) from the *Type* drop down list and change the inflection points ($X1/L$ & $X3/L$) to **zero**, since we assumed a parabola with no inflection points. Keep the low point ($X2/L$) at mid-span, i.e., at **0.5**.

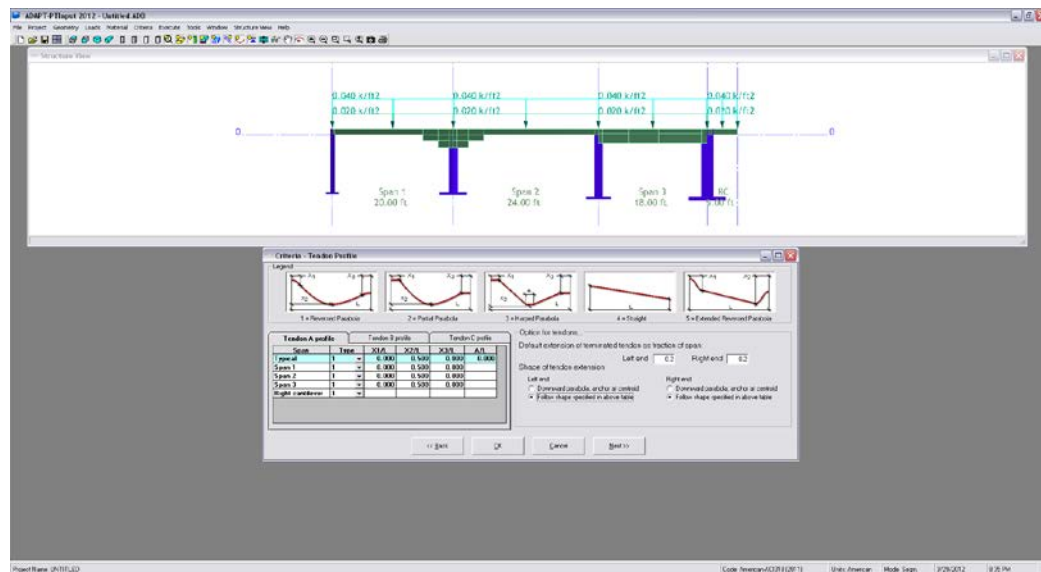


FIGURE 5.3-25

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Minimum Covers*.

5.3.6 Specify Minimum Covers For Post-Tensioning Tendons And Mild Steel Reinforcement (Fig. 5.3-26)

The cover for the prestressing steel is specified to the center of gravity of the strand (cgs). Therefore, for ½ inch (13 mm) strand, cgs is minimum cover + ½ * ½ ,i.e., cgs = cover + 0.25”(cgs = cover + ½ * 13). Keep the **default values** for both Post-tensioning and Non-prestressed Reinforcement.

Note that the default value for cgs at the exterior span is set to 1.75”. If exterior spans are considered “unrestrained,” with respect to fire resistivity, they require a higher cover.

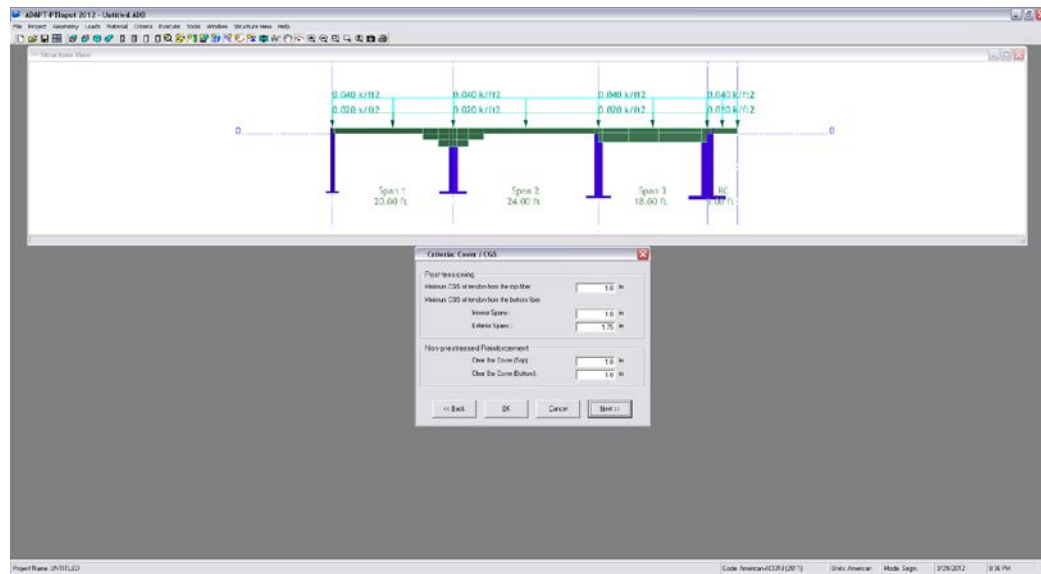


FIGURE 5.3-26

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Minimum Bar Extension*.

5.3.6.1 Specify Minimum Bar Length and Bar Extension Of Mild Steel Reinforcement (Fig. 5.3-27)

The values given are for development of bars required to supplement prestressing in strength check. Modify if necessary. For this tutorial use the default values.

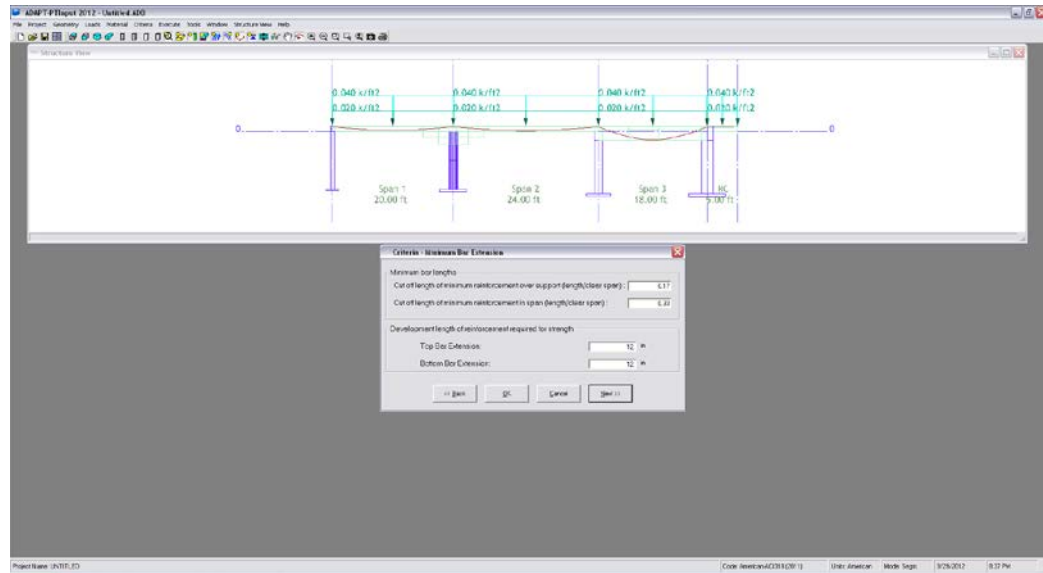


FIGURE 5.3-27

Click **Next** at the bottom of the screen to open the next input screen, *Load Combinations*.

5.3.6.2 Input Load Combinations (Fig. 5.3-28, 29, 30)

This screen is used to input the load combination factors for service and strength (ultimate) load conditions. It is also used to enter any applicable strength reduction factors. The default values are according to the ACI 318-2011. For this example, use the default values.

The program allows you to specify four strength load combinations and four service load combinations. For ACI 318-2011, two of the service load combinations are reserved for sustained load and two for total load.

ADAPT-PT 2012 allows lateral moments to be included and designed for in combination with gravity loads. To do this, select the check mark to *Include lateral loads* and click on the *Set Values* button to define *Lateral moments* (**Fig. 5.3-21**) and *Lateral load combinations* (**Fig. 5.3-22**). For this example, lateral loads will not be included.

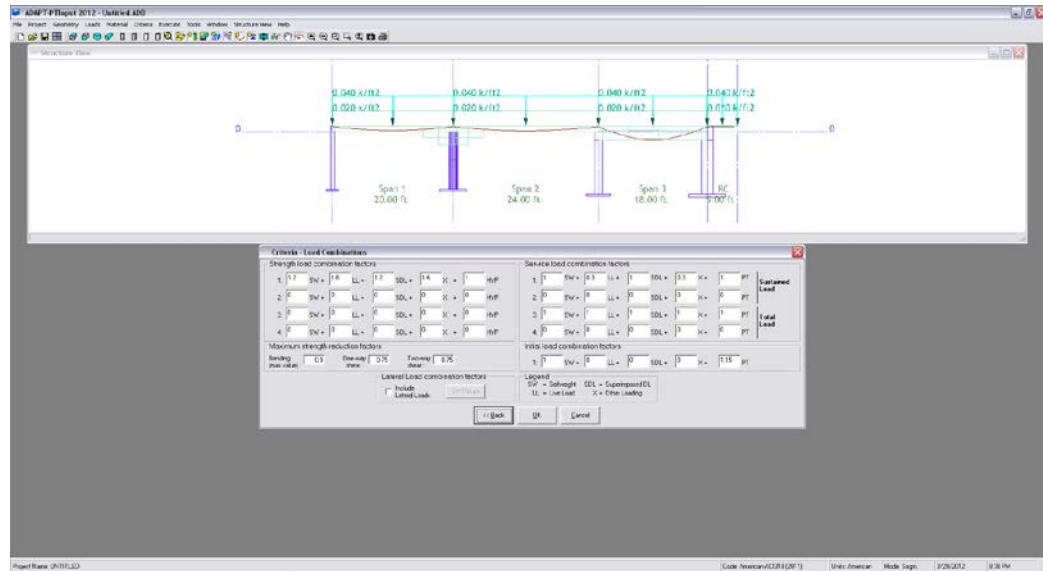


FIGURE 5.3-28

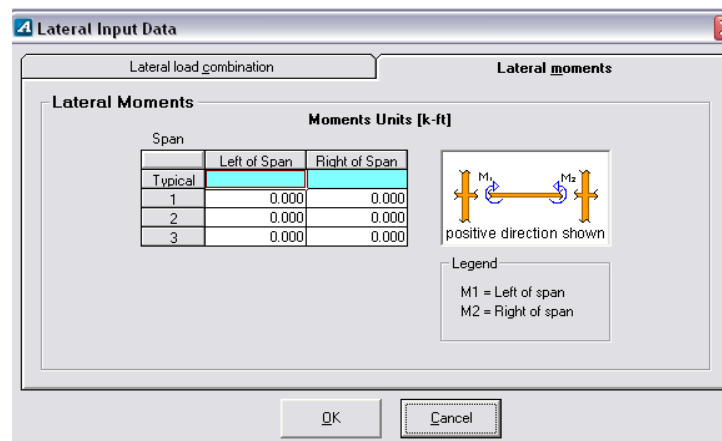


FIGURE 5.3-29

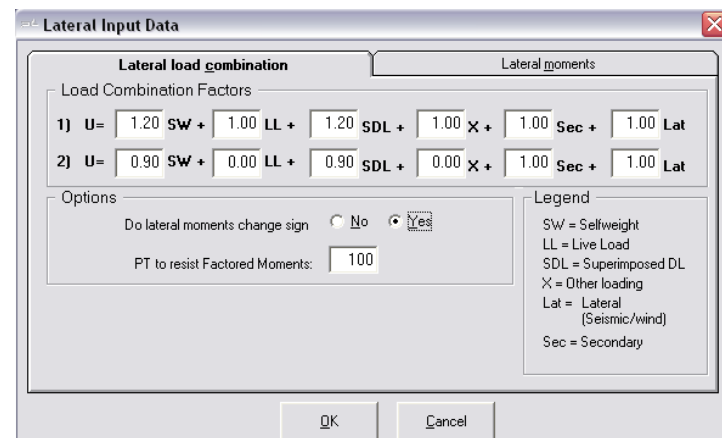



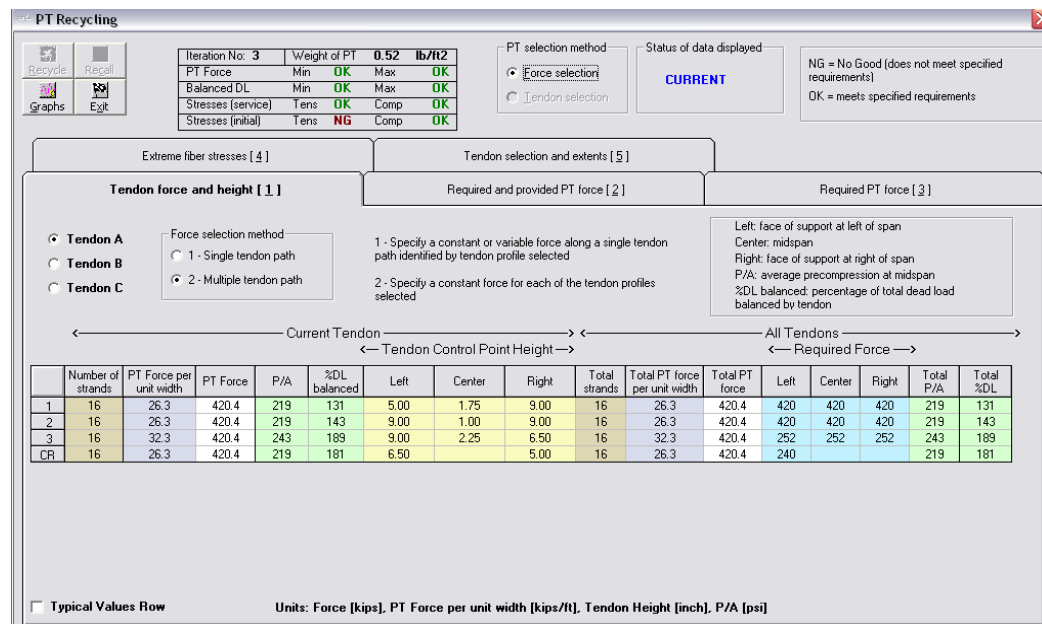
FIGURE 5.3-30

Click **OK** at the bottom of the screen to finish the input wizard.

5.4 SAVE AND EXECUTE THE INPUT DATA

To save the input data and execute the analysis, either select **Execute Analysis** from the *Action* menu of the menu bar or click on the **Save & Execute Analysis** button . Then, give a **file name** and **directory** in which to save the file. The program saves all sub-files in a specific folder with the name selected by the user, along with the .adb file at the user-defined directory. Once the file is saved, the program will automatically execute the analysis by reading the data files and performing a number of preliminary data checks.

Once the execution completes the selection of post-tensioning, the “PT Recycling” window, as shown in **Figure 5.4-1** opens. If an error is detected, the program will stop and display a message box indicating the most likely source of the error.



PT Recycling

Iteration No: **3** Weight of PT: **0.52 lb/ft²**

PT Force	Min	OK	Max	OK
Balanced DL	Min	OK	Max	OK
Stresses (service)	Tens	OK	Comp	OK
Stresses (initial)	Tens	NG	Comp	OK

PT selection method: ☒ Force selection ☐ Tendon selection

Status of data displayed: **CURRENT**

NG = No Good (does not meet specified requirements)
OK = meets specified requirements

Extreme fiber stresses [4] Tendon selection and extents [5]

Tendon force and height [1]

☒ Tendon A ☐ Tendon B ☐ Tendon C

Force selection method:
☐ 1 - Single tendon path
☒ 2 - Multiple tendon path

1 - Specify a constant or variable force along a single tendon path identified by tendon profile selected
 2 - Specify a constant force for each of the tendon profiles selected

Left: face of support at left of span
 Center: midspan
 Right: face of support at right of span
 P/A: average precompression at midspan
 %DL balanced: percentage of total dead load balanced by tendon

Current Tendon: All Tendons

← Tendon Control Point Height → ← Required Force →

	Number of strands	PT Force per unit width	PT Force	P/A	%DL balanced	Left	Center	Right	Total strands	Total PT force per unit width	Total PT force	Left	Center	Right	Total P/A	Total %DL
1	16	26.3	420.4	219	131	5.00	1.75	9.00	16	26.3	420.4	420	420	420	219	131
2	16	26.3	420.4	219	143	9.00	1.00	9.00	16	26.3	420.4	420	420	420	219	143
3	16	32.3	420.4	243	189	9.00	2.25	6.50	16	32.3	420.4	252	252	243	189	
CR	16	26.3	420.4	219	181	6.50		5.00	16	26.3	420.4	240			219	181

☐ Typical Values Row Units: Force [kips], PT Force per unit width [kips/ft], Tendon Height [inch], P/A [psi]

FIGURE 5.4-1

Here you can optimize the design by changing the tendon forces and tendon heights.

Change the tendon force and heights as shown in **Figure 5.4-2**. The status indicator at the top right of the Recycle window will begin to flash.

Since we selected “Force Selection” option during data entry, the program will only allow the “Force Selection” mode for execution.

Once all of the changes are made, click on the **Recycle** button to update all of the tabs, the “Design Indicator” box and the “Recycle Graphs”. There is no limit on

the number of changes that can be made or the number of times the window can be recycled.

PT Recycling

Iteration No: **3** Weight of PT: **0.52 lb/ft²**

PT Force: Min --- Max ---
 Balanced DL: Min --- Max ---
 Stresses (service): Tens --- Comp ---
 Stresses (initial): Tens --- Comp ---

PT selection method:
☒ Force selection
☐ Tendon selection

Status of data displayed:
RECYCLE

NG = No Good (does not meet specified requirements)
 OK = meets specified requirements

Extreme fiber stresses [4] Tendon selection and extents [5]

Tendon force and height [1] Required and provided PT force [2] Required PT force [3]

Tendon A
 Force selection method:
☒ 1 - Single tendon path
☐ 2 - Multiple tendon path

1 - Specify a constant or variable force along a single tendon path identified by tendon profile selected
 2 - Specify a constant force for each of the tendon profiles selected

Left: face of support at left of span
 Center: midspan
 Right: face of support at right of span
 P/A: average precompression at midspan
 %DL balanced: percentage of total dead load balanced by tendon

Current Tendon All Tendons

	Number of strands	PT Force per unit width	PT Force	P/A	%DL balanced	Left	Center	Right	Total strands	Total PT force per unit width	Total PT force	Left	Center	Right	Total P/A	Total %DL
1	9	15.0	240.0	125	75	5.00	1.75	9.00	9	15.0	240.0	420	420	420	125	75
2	9	15.0	240.0	125	82	9.00	1.00	9.00	9	15.0	240.0	420	420	420	125	82
3	9	17.3	225.0	130	106	9.00	2.25	7.00	9	17.3	225.0	252	252	252	130	106
CR	9	15.0	240.0	125	138	7.00		5.00	9	15.0	240.0	240			125	138

☐ Typical Values Row Units: Force [kips], Tendon Height [inch], P/A [psi]

FIGURE 5.4-2

After the recalculation of the stresses and required forces along the member based on the current values, the window, as shown in **Figure 5.4-3** with “Current” status indicator, opens and all the indicators show “OK”.

PT Recycling

Iteration No: **4** Weight of PT: **0.30 lb/ft²**

PT Force: Min **OK** Max **OK**
 Balanced DL: Min **OK** Max **OK**
 Stresses (service): Tens **OK** Comp **OK**
 Stresses (initial): Tens **OK** Comp **OK**

PT selection method:
☒ Force selection
☐ Tendon selection

Status of data displayed:
CURRENT

NG = No Good (does not meet specified requirements)
 OK = meets specified requirements

Extreme fiber stresses [4] Tendon selection and extents [5]

Tendon force and height [1] Required and provided PT force [2] Required PT force [3]

Tendon A
 Force selection method:
☒ 1 - Single tendon path
☐ 2 - Multiple tendon path

1 - Specify a constant or variable force along a single tendon path identified by tendon profile selected
 2 - Specify a constant force for each of the tendon profiles selected

Left: face of support at left of span
 Center: midspan
 Right: face of support at right of span
 P/A: average precompression at midspan
 %DL balanced: percentage of total dead load balanced by tendon

Current Tendon All Tendons

	Number of strands	PT Force per unit width	PT Force	P/A	%DL balanced	Left	Center	Right	Total strands	Total PT force per unit width	Total PT force	Left	Center	Right	Total P/A	Total %DL
1	9	15.0	240.0	125	75	5.00	1.75	9.00	9	15.0	240.0	420	420	420	125	75
2	9	15.0	240.0	125	82	9.00	1.00	9.00	9	15.0	240.0	420	420	420	125	82
3	9	17.3	225.0	130	106	9.00	2.25	7.00	9	17.3	225.0	252	252	252	130	106
CR	9	15.0	240.0	125	138	7.00		5.00	9	15.0	240.0	240			125	138

☐ Typical Values Row Units: Force [kips], Tendon Height [inch], P/A [psi]

FIGURE 5.4-3

You can check the final stresses either by clicking **Extreme fiber stresses [4]** tab in the *PT Recycling* window (**Fig. 5.4-3**) or by clicking **Graphs** at the top left of the screen.

Graphs displays a set of three graphs which provide detailed information on the tendon profile, the tension and compression stresses and the required versus provided post-tensioning forces at 1/20th points along the spans (**Fig. 5.4-4**).

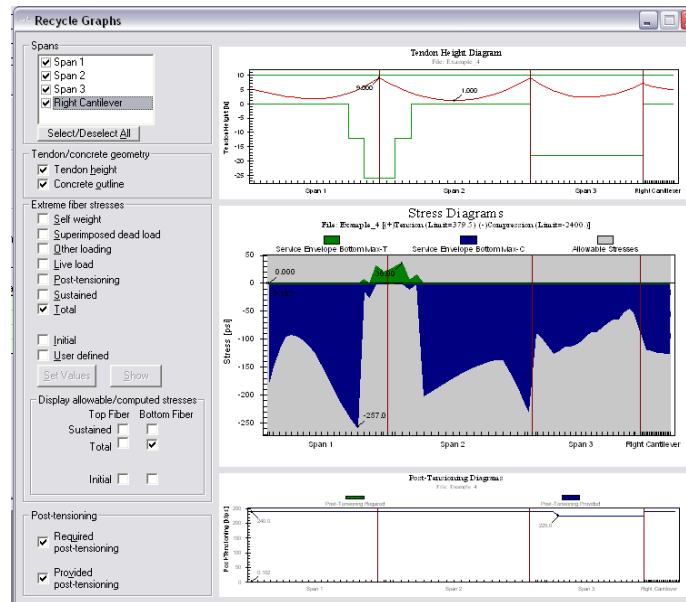


FIGURE 5.4-4

The top diagram, the **Tendon Height Diagram** shows the elevation of the tendon profile selected. The tendon profile can be viewed either with or without the concrete outline by checking the option at the left of the screen.

The second diagram, **Stress Diagrams**, plots the maximum compressive and tensile stresses at the top and bottom face of the member. You can view the stresses due to *Dead Load*, *Live Load*, *Post-tensioning* and *Service Combination* each separately, or in combination, by selecting the options at the screen. Also you can verify the top and bottom stresses due to service combination with the allowable values. In **Figure 5.4-4**, it shows the final bottom fiber stress with allowable. In which, gray color represents the *allowable stresses*, top curve represents the *tensile stress* and bottom curve represents the *compressive stress*. If the calculated stress is not within the limit, i.e., the top or bottom curve is outside the gray portion; you need to modify the forces to optimize the design.

The third diagram, **Post-Tensioning Diagrams** shows the required and provided post-tensioning force at 1/20th points along each span. The *vertical line* represents the *required* post-tensioning and the *horizontal line* represents the *provided* post-

tensioning at that section. In the **Figure 5.4-4** note that in most parts of the structure the required post-tensioning is zero. That is to say, the stresses due to service combination of dead and live load do not exceed the allowable tension of the code. However, according to the code a minimum precompression of **125 psi (0.86 MPa)** is provided (**Fig. 5.4-5**). At each design section along a span, the program performs an analysis based on the post-tensioning force at that section.

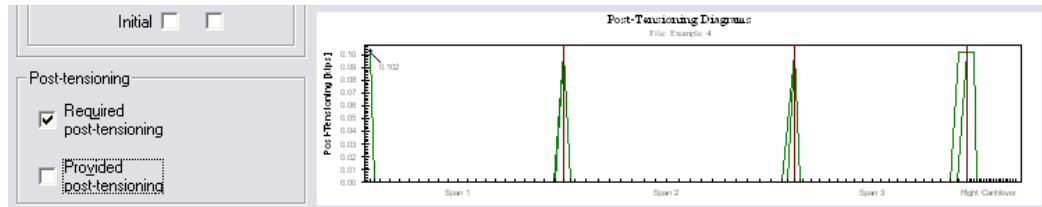


FIGURE 5.4-5

If the solutions are not acceptable to you, you can change the post-tensioning layout and recycle until an acceptable solution is reached. Once you are satisfied with the solution, select **Exit** at the top left of the screen to continue with the calculations.

The program continues with the calculations based on the most recent tendon forces and profile selection. Once finished successfully, you return to the main program window with the screen as shown in **Figure 5.4-6**.

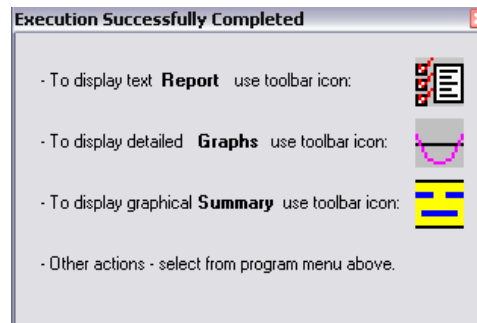



FIGURE 5.4-6

Close the above window by clicking **X** at the top right corner.

5.5 CREATE REPORTS

PT 2012 includes a Report Generator allowing the user to create full tabular, graphical reports or to customize any report according to predetermined report sections. To setup the report, select the **Report Setup** item on the *Options* menu or click the **Report Setup** button  on the main toolbar. The Report Generator screen shown in **Figure 5.5-1** will open.

The program allows you to generate reports in an MS-Word® editable format. You have the following options as explained below:

- Report cover: Select this option to generate a report cover with your logo and company information. To update your company information, click on **Update Company Info** on the *Report Generator* and you will see the screen **Company Information** shown in **Figure 5.5-2**.
- Table of Contents
- Concise Report: This report includes Project Design Parameters and Load Combinations as well as a Design Strip Report containing Geometry, Applied Loads, Design Moments, Tendon Profile, Stress check / Code check, Rebar Report, Punching Shear and Deflections. The program now reports Material Quantities in this report.
- Tabular Reports – Compact
- Tabular Reports – Detailed: This report now includes Demand Moments and Moment Capacities
- Graphical Reports
- Legend

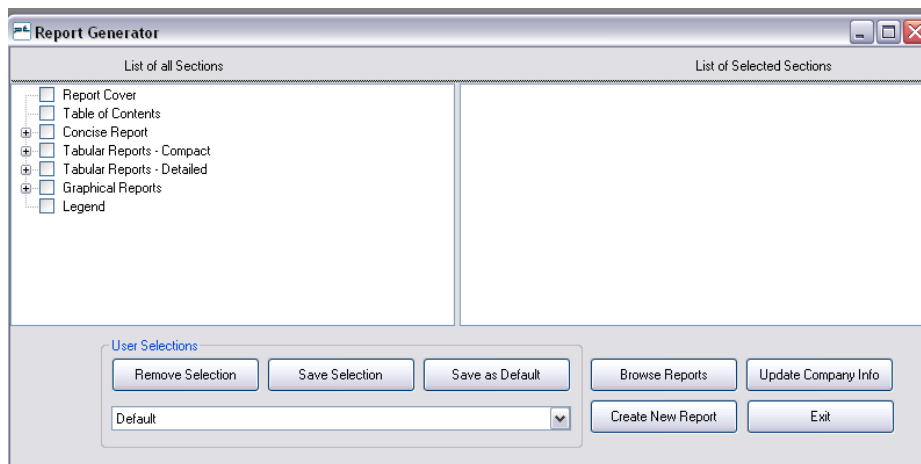


FIGURE 5.5-1

Simply check any item in the *List of all Sections* to include it in the report. The item will then appear in the *List of Selected Sections* on the right hand side of the *Report Generator*.

To generate and view the report, click on **Generate/View Report** on the bottom of the *Report Generator*.


The program allows you to open and view existing reports by clicking on **Open Reports**.

The Report Generator allows you to save report content as either a default template or as a user defined template. This enables you to quickly select content for any project by either using the default content or any other user defined content.

To define content as the default template, select report content from the List of all Sections and click on **Save as Default**.

To define content as a user defined template, select report content from the List of all Sections and click on **Save Selection**. You are asked to enter a name for your selection. This name appears then in the drop down box in the **User Selections** frame.

FIGURE 5.5-2

To open the “PT Summary Report” (**Fig. 5.5-3**) either click the **PTSum** button  on the tool bar or select the **PT summary** item on the *View* menu.

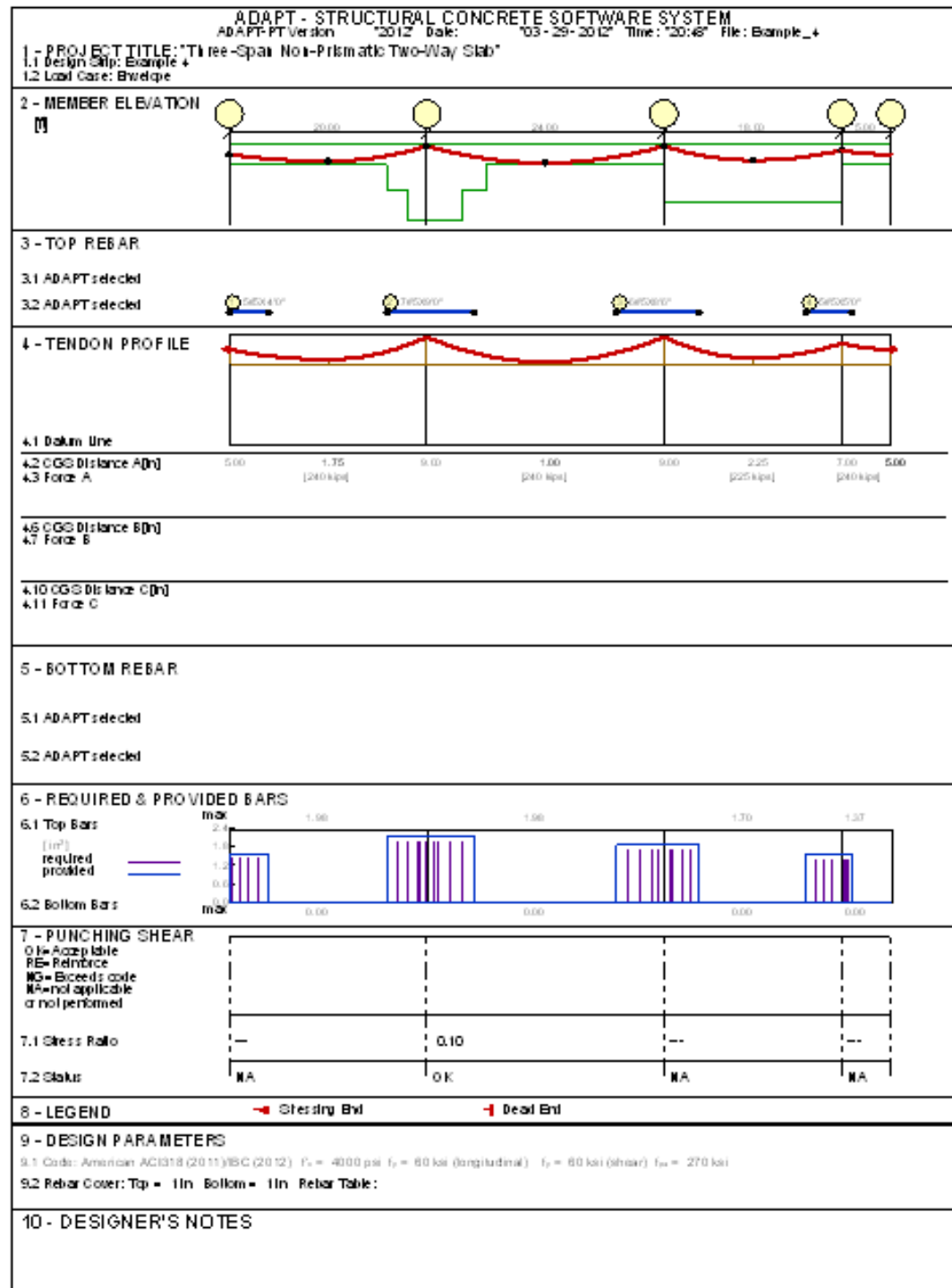


FIGURE 5.5-3

To view the graphs, either click the **Show Graphs** button  from the toolbar or select **Graphs** on the View menu.